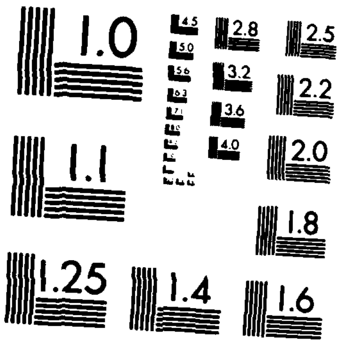
AD-A129 025 MAGNA (MATERIALLY AND GEOMETRICALLY NONLINEAR ANALYSIS) 1/3
PART II PREPROCES..(U) DAYTON UNIV OH RESEARCH INST
T S BRUNER ET AL. DEC 82 UDR-TR-82-112
UNCLASSIFIED AFWAL-TR-82-3098-PT-2 F33615-80-C-3403 F/G 9/2 NL


1.0
4.5
2.8
2.5
5.0
5.6
3.2
2.2
6.3
3.6
4.0
1.1
2.0
1.8
1.25
1.4
1.6

AFWAL-TR-82-3098, Part II

(This report supersedes AFWAL-TR-81-3180, dated February 1982)

MAGNA (Materially and Geometrically Nonlinear Analysis

Part II - Preprocessor Manual

T. S. Bruner
R. A. Brockman
K. A. Primrose

University of Dayton Research Institute
300 College Park Avenue
Dayton, Ohio 45469

Final Report, March 1980 - December 1982

December 1982

Approved for public release; distribution unlimited.

FLIGHT DYNAMICS LABORATORY
AIR FORCE WRIGHT AERONAUTICAL LABORATORIES
AIR FORCE SYSTEMS COMMAND
WRIGHT-PATTERSON AIR FORCE BASE, OHIO 45433

NOTICE

When Government drawings, specifications, or other data are used for any purpose other than in connection with a definitely related Government procurement operation, the United States Government thereby incurs no responsibility nor any obligation whatsoever; and the fact that the government may have formulated, furnished, or in any way supplied the said drawings, specifications, or other data, is not to be regarded by implication or otherwise as in any manner licensing the holder or any other person or corporation, or conveying any rights or permission to manufacture, use, or sell any patented invention that may in any way be related thereto.

This report has been reviewed by the Office of Public Affairs (ASD/PA) and is releasable to the National Technical Information Service (NTIS). At NTIS, it will be available to the general public, including foreign nations.

This technical report has been reviewed and is approved for publication.

Robert E. McCarty

ROBERT E. McCARTY
Project Engineer

Charles A. Babish III

CHARLES A. BABISH, III
Acting Group Leader
Subsystems Development Group

FOR THE COMMANDER

SOLOMON R. METRES
Director
Vehicle Equipment Division

"If your address has changed, or if you wish to be removed from our mailing list, or if the addressee is no longer employed by your organization please notify AFWAL/FIER, W-PAFB OH 45433 to help us maintain a current mailing list".

Copies of this report should not be returned unless return is required by security considerations, contractual obligations, or notice on a specific document.

UNCLASSIFIED

SECURITY CLASSIFICATION OF THIS PAGE (When Data Entered)

REPORT DOCUMENTATION PAGE		READ INSTRUCTIONS BEFORE COMPLETING FORM
1. REPORT NUMBER AFWAL-TR-82-3098, Part II	2. GOVT ACCESSION NO.	3. RECIPIENT'S CATALOG NUMBER
4. TITLE (and Subtitle) MAGNA (Materially and Geometrically Nonlinear Analysis) Part II - Preprocessor Manual		5. TYPE OF REPORT & PERIOD COVERED Final Report March 1980 - Dec. 1982
		6. PERFORMING ORG. REPORT NUMBER UDR-TR-82-112
7. AUTHOR(s) T. S. Bruner R. A. Brockman K. A. Primrose		8. CONTRACT OR GRANT NUMBER(s) F33615-80-C-3403
9. PERFORMING ORGANIZATION NAME AND ADDRESS University of Dayton Research Institute 300 College Park Avenue Dayton, Ohio 45469		10. PROGRAM ELEMENT, PROJECT, TASK AREA & WORK UNIT NUMBERS Prog. Ele. 62201F Proj. 2402, Task 240203 Work Unit 24020332
11. CONTROLLING OFFICE NAME AND ADDRESS Flight Dynamics Laboratory (AFWAL/FIER) Air Force Wright Aeronautical Laboratories Wright-Patterson Air Force Base, OH 45433		12. REPORT DATE December 1982
		13. NUMBER OF PAGES 222
14. MONITORING AGENCY NAME & ADDRESS(if different from Controlling Office)		15. SECURITY CLASS. (of this report) Unclassified
		15a. DECLASSIFICATION/DOWNGRADING SCHEDULE

16. DISTRIBUTION STATEMENT (of this Report)

Approved for public release; distribution unlimited

DTIC ELECTE JUN 6 1983 S H

17. DISTRIBUTION STATEMENT (of the abstract entered in Block 20, if different from Report)

18. SUPPLEMENTARY NOTES

The computer program contained in this technical report is theoretical and in no way affects any Air Force owned software programs.

19. KEY WORDS (Continue on reverse side if necessary and identify by block number)

Finite Element Analysis
Computer Graphics
Preprocessing
Mesh Generation
Aircraft Transparency
Surface Modeling

20. ABSTRACT (Continue on reverse side if necessary and identify by block number)

A preprocessor system developed for use in connection with three-dimensional finite element analysis is described. The system consists of several data entry modules, a central preprocessor and various data translation and conversion utilities. Data entry and generation functions include keyboard, digitizer, and file inputs, which are oriented toward the type of thick shell geometry currently encountered in aircraft

DD FORM 1473 1 JAN 73 EDITION OF 1 NOV 65 IS OBSOLETE

UNCLASSIFIED

SECURITY CLASSIFICATION OF THIS PAGE (When Data Entered)

20. (Concluded)

transparency analysis. The main preprocessor operates on models in "superelement" form, permitting selective mesh refinement, numerous geometric transformations, and merging of separate models to form the final analysis mesh. Data reformatting modules permit the generation of a complete input file for finite element analysis, and/or the translation of finite element data into other forms for archiving or further processing external to the preprocessor itself.

FOREWORD

This report describes the finite element solution program MAGNA, developed at the University of Dayton Research Institute, Dayton, Ohio. Development of the program was performed between January, 1978 and December, 1982, by the Analytical Mechanics Group (Dr. F. K. Bogner, Leader) within the Aerospace Mechanics Division (D. H. Whitford, Supervisor) of the Research Institute.

This work effort was accomplished under Project 2402, "Vehicle Equipment Technology," Task 240203, "Aerospace Vehicle Recovery and Escape Subsystems," Work Unit 24020332, "Computer Aided Design of Bird-Resistant Transparencies for USAF Aircraft."

The present report provides final documentation of the developments performed on Air Force Contract F33615-80-C-3403 between March, 1980 and December, 1982 for the Flight Dynamics Laboratory, Air Force Wright Aeronautical Laboratories, Wright-Patterson Air Force Base, Ohio. The project manager for this effort was Dr. Fred K. Bogner, and the Principal Investigator was Dr. Robert A. Brockman. Technical direction and support was provided by Mr. Robert E. McCarty (AFWAL/FIER) as the Air Force Project Engineer. The work described herein represents a continuation of previous developments performed in-house at the University of Dayton Research Institute, and on Air Force Contract F33615-76-C-3103.

The author wishes to express his appreciation for the contributions of several individuals and organizations whose efforts, support, and suggestions have resulted in significant improvements to the MAGNA program. Continuing support and many useful discussions have been provided by Dr. Fred K. Bogner; numerous improvements to both the program and its documentation have been suggested by Mr. Robert E. McCarty. The analytical development performed by Dr. H. C. Rhee and Dr. Mohan L. Soni, and the computer graphics support provided by Messrs. T. S. Bruner, C. S. King, M. P. Bouchard, M. J. Hecht, Ms. M. A. Dominic, and Ms. M. E. Wright are also gratefully acknowledged. Mr. Thomas W. Held performed the conversion of MAGNA to the VAX 11/780. Computer resources and assistance in adapting the program to the CRAY-1 computer were provided by United Information Services; special thanks are due to Mr. Kent Griffith of UIS, who developed the necessary direct access file utilities. Finally, the efforts of Ms. Kathy Reineke in typing the manuscript of this manual are deeply appreciated.

This report (Parts I, II, III, and IV) supersedes AFWAL-TR-80-3152, AD A099454 dated January 1981; AFWAL-TR-80-3151, AD A099530 dated January 1981; AFWAL-TR-81-3180, AD A117544 dated February 1982; and AFWAL-TR-81-3181, AD A116541 dated February 1982.

TABLE OF CONTENTS

SECTION		PAGE
1	INTRODUCTION	1.1
	1.1 Overview of the Preprocessor Functions	1.2
	1.2 Paths Through the System	1.5
2	DATA ENTRY FUNCTIONS	2.1
	2.1 CREATE - Coarse Mesh Input	2.3
	2.2 IJKGEN - Analytical Surface Geometry Input	2.4
	2.3 CORGEN - General Surface Geometry Input	2.15
	2.3.1 Interactive Program Execution	2.22
	2.3.2 COMMands Listing	2.25
	2.3.3 COORdinate System	2.27
	2.3.4 DIGItizing Data Command	2.27
	2.3.5 DISPlay Values	2.39
	2.3.6 ELEMent Data	2.39
	2.3.7 FINIsh Model	2.42
	2.3.8 GROUp Data	2.42
	2.3.9 INTErpolate Data	2.43
	2.3.10 NODE Data	2.46
	2.3.11 RESTore Data	2.49
	2.3.12 SAVE Data	2.49
	2.3.13 STOP Program	2.50
	2.4 AGRID - Arbitrary Arrays of Points Mesh Input	2.51
	2.5 SPATCH - Surface Patch Geometry Input	2.57
3	PREPROCESSOR INTERFACE MODULES	3.1
	3.1 TRNSFR - MAGNA Input Data Translator	3.2
	3.2 IMPRINT - IMPRESS Preprocessor Interface	3.3
	3.3 General Purpose Interface Routines	3.5
4	MODEL VERIFICATION AND MODIFICATION	4.1
	4.1 Modification/Refinement of Model Geometry	4.3
	4.2 Element Properties Selection	4.18
	4.3 Boundary Conditions and Constraints	4.20

TABLE OF CONTENTS (continued)

SECTION		PAGE
	4.4 Applied Loading	4.21
	4.5 Node Point Reordering	4.23
	4.6 Plotting Utilities	4.26
	4.7 Data Listing Utilities	4.29
	4.8 Typical Sequences of Operations	4.30
5	MODEL DATA TRANSLATION	5.1
	5.1 REFMT - MAGNA Data Translator	5.2
	5.2 NEUTRAL - Neutral File Translation Program	5.3
6	MODEL DATA FILES	6.1
	6.1 Preprocessor Internal Data Structure	6.2
	6.1.1 Shell Surface Geometry File	6.2
	6.1.2 Three-Dimensional Model File	6.3
	6.2 Neutral Data File Structure	6.4
	6.3 MAGNA Input Data File	6.5
7	DEMONSTRATION PROBLEMS	7.1
	7.1 Execution of CREATE / PREP / REFMT Sequence	7.1
	7.1.1 Summary of Modeling Procedure	7.2
	7.1.2 CREATE Execution	7.4
	7.1.3 PREP Execution	7.6
	7.1.4 REFMT Execution	7.21
	7.2 Execution of CORGEN / EXPAND / PREP / REFMT Sequence	7.24
	7.2.1 Summary of Modeling Procedure	7.24
	7.2.2 CORGEN Execution	7.28
	7.2.3 EXPAND Execution	7.31
	7.2.4 PREP Execution	7.32
	7.2.5 REFMT Execution	7.49
	7.3 Execution of IJKGEN / PREP / REFMT Sequence	7.51
	7.3.1 Summary of Modeling Procedure	7.51
	7.3.2 IJKGEN Execution	7.59

TABLE OF CONTENTS (continued)

SECTION		PAGE
	7.3.3 PREP Execution	7.59
	7.3.4 REFMT Execution	7.73
8	REFERENCES	8.1
9	APPENDICES	A.1
	APPENDIX A. Preprocessor File Descriptions	A.2
	APPENDIX B. Possible Data Paths	A.6
	APPENDIX C. Access Procedures for all Modules	A.9
	APPENDIX D. User Subroutines for IJKGEN	A.12
	APPENDIX E. User Subroutines for SPATCH	A.15
	APPENDIX F. Material Properties Library	A.19

LIST OF FIGURES

NUMBER		PAGE
1.1	Preprocessor Organization	1.6
2.1	27 Node Solid Element	2.5
2.2	Four and Nine Node Surface Elements	2.6
2.3	Types of Geometries Suitable for IJKGEN	2.7
2.4	Example of IJKGEN Method of Generation	2.9
2.5	IJKGEN Default Coordinate Options	2.10
2.6	IJKGEN Default Mesh Spacing Options	2.11
2.7	Sample Execution of IJKGEN	2.13
2.8	Plot of IJKGEN Sample Model from PREP	2.14
2.9	Portion of B-1 Bomber Windshield	2.16
2.10	Coarse B-1 Windshield Mesh using Key Point Coordinates	2.17
2.11	Lofting Surface	2.19
2.12	Stineman Interpolation	2.21
2.13	CORGEN Default Options and Commands	2.24
2.14	CORGEN Command/Menu Paths	2.24
2.15	Master Commands for CORGEN	2.26
2.16	Right Handed Cartesian (X,Y,Z) Coordinate System; Right Handed Spherical (R,ϕ,θ) Coordinate System; Right Handed Cylindrical (R,θ,Z) Coordinate System	2.28
2.17	DIGItize Command Initialization Questions	2.30
2.18	DIGItize Command Menu Block	2.32
2.19	CORGEN Default Lofting Plane	2.35
2.20	DIGItize Command HELP Feature	2.37
2.21	Biquadratic Surface Element Generated in CORGEN	2.38
2.22	DISPlay Command Output Listing	2.40
2.23	ELEMent Command Options	2.41
2.24	GROUP Command Options	2.44
2.25	NODE Command Options	2.47
2.26	Arbitrary Array of Surface Points	2.52
2.27	AGRID Interpolation Grid	2.53
2.28	Boundary Definition for Output Grid	2.54
4.1	FILL and MASK Operations	4.11
4.2	Effect of the MERGE Command	4.12

LIST OF FIGURES, continued

NUMBER		PAGE
4.3	Effect of the REFINE Command	4.13
4.4	REFLECT, ROTATE and TRANSLATE Options	4.14
4.5	Effect of the REFINE Operation Prior to FILL	4.15
4.6	Surface Numbers for 3-D Pressure Loads	4.24
4.7	Edge Numbers for Line Loads for Axisymmetric Pressures	4.25
4.8	Example of Node Point Numbering Optimization	4.27

LIST OF TABLES

NUMBER		PAGE
1.1	Summary of Input Data Types for Data Entry Modules	1.7
2.1	CORGEN Interactive Processing Capabilities	2.23
2.2	CORGEN Group Designators	2.45
2.3	AGRID Data Input	2.56
3.1	IMPRESS Data Types	3.4
3.2	INICNV - Initialization Subroutine	3.6
3.3	NODCNV - Nodal Coordinate Conversion Routine	3.7
3.4	ELECNV - Element Data Conversion Routine	3.8
3.5	BCDCNV - Boundary Condition Conversion Routine	3.9
3.6	NLDCNV - Nodal Loads Conversion Routine	3.10
3.7	ELDCNV - Element Loads Conversion Routine	3.11
3.8	TRMCNV - Termination Subroutine	3.12
4.1	The FILL Command	4.4
4.2	The MASK Command	4.5
4.3	The MERGe Command	4.6
4.4	The REFIne Command	4.7
4.5	The REFLect Command	4.8
4.6	The ROTAte Command	4.9
4.7	The TRANslate Command	4.10
4.8	Alphabetized Listing of PREP Commands	4.31
A.1	Shell Surface Model Data File	A.3
A.2	Three-Dimensional Model Data File	A.4
A.3	Neutral Model File Data Format	A.5
B.1	Preprocessor Module Functions	A.7
B.2	Preprocessor File Types	A.8
C.1	CDC Access to MAGNA Preprocessing System	A.10
C.2	VAX Access to MAGNA Preprocessing System	A.11

SECTION 1

INTRODUCTION

Finite element methods and other numerical techniques for performing complex structural analyses have matured to the point that they may be used with confidence in the development and final qualification of complex structural designs. Numerical structural analysis is routinely used in a number of industries to ensure maximum safety and reliability, and practical applications of enormous size have been accomplished [1] for the qualification of designs for which full-scale testing is impractical or impossible.

As the size and/or complexity of an application increases, however, the advantages of a computer simulation are sometimes reduced due to the time and cost associated with the preparation and checking of input data. For the geometric description of a finite element model, the tabulation of nodal point coordinates and the connections of elements to the nodes accounts for the bulk of the numerical data. In a large finite element model, manual preparation of this data alone may consume weeks or even months.

This report describes a system of data preprocessing programs which generate and manipulate modeling input for three-dimensional finite element structural analysis. The preprocessing system is oriented toward the construction of models for applications involving thick shell and solid geometries, although axisymmetric and two-dimensional models may also be prepared. Most of the facilities of the preprocessor are concerned with the geometric description of a structure, since it represents the largest and most complex portion of the problem definition. However, the remaining parts of the model, such as constraints, loading, and material properties, can also be prepared automatically. Facilities are also provided within the present system of programs for interactive plotting, optimization of a completed model for solution efficiency, and communication with other computer programs which perform related data processing functions.

1.1 OVERVIEW OF PREPROCESSOR FUNCTIONS

The present preprocessing system is composed of a series of compatible computer programs, which perform three primary classes of functions:

- data entry and translation,
- model editing, refinement and display, and
- data output and reformatting.

These functions are outlined briefly below, and in detail in Sections 2, 3 and 4 respectively.

The motivation for developing the preprocessor described here is the current need for a means of constructing finite element analysis models for aircraft windshield-type structures, which can be broadly characterized as solid or thick shell constructions. As a result, most input functions are strongly oriented toward problems involving curved, three-dimensional geometries which might be described by surface shapes and thickness distributions, by lofting coordinates, or by mathematical expressions in curvilinear coordinates.

Data entry functions performed within the preprocessor provide a convenient means of defining the geometry of this thick shell / solid class of structures; analytically-defined shapes, lofting input, coarse surface grid data, and surface patch input are supported, by means of file, keyboard and/or digitizing pad input. "Superelement" input, which defines a geometric shape in terms of the coarsest possible finite element mesh, is also possible for general two- and three-dimensional regions. For models prepared using other data preprocessing systems (such as PATRAN-G [2] or IMPRESS [3]), data translation routines are available for converting most finite element data from external formats to the data forms understood by this system.

Every data entry (or translation) function results in data files, which are stored in a certain internal format for use in the preprocessor. This format is designed for easy translation to other forms as needed to interface with other preprocessors and/or specific analysis programs.

Once a finite element model is stored in the internal format of the preprocessor, any number of operations may be performed to edit, refine or otherwise manipulate the model geometry and properties. Two or more models may be merged to form a single geometric model at this stage as well. During this phase of preprocessing, a model can always be saved and accessed later for further processing, so that the modelling process need not be accomplished in a single session at the computer terminal.

Final version(s) of a completed finite element model are translated into forms understood by other computer programs in the data output and reformatting process. The finite element data can be formatted for a specific analysis program, or rewritten in a form which can be transferred to other preprocessor formats as needed.

The interrelation of these three phases of preprocessing is discussed in detail in the following subsection, which describes many of the possible data paths through the total system. Since the preprocessing system is constructed from a group of separate programs which may be executed in any order, the possibility of introducing specially-written small programs to perform unusual or unsupported functions also exists. This open-ended organization can be a great advantage for the experienced user who is handy with FORTRAN, while less experienced (or less computer-oriented) individuals can exercise the capabilities of the system "as-is".

1.2 PATHS THROUGH THE SYSTEM

The organization of the preprocessor is shown in its most general form in Figure 1.1. Geometric and other data are entered via keyboard, data file or digitizing pad, using one of the data entry or interface facilities of the system. At the opposite end, a completed model is extracted from the system format and placed in a usable form using additional data translators. In between, the main preprocessor (PREP) may be used to modify, refine, list, plot or otherwise manipulate the data.

The choice of a data entry program is invariably governed by the form in which the information describing the model is most readily accessible. Table 1.1 summarizes the types of input which are accepted by each of the data entry utilities : CREATE, IJKGEN, CORGEN, AGRID, and SPATCH.

In some situations, the geometric description of a model may already exist in the form of a finite element mesh. When this is the case, one of the data interfaces TRNSFR, IMPRINT, or NEUTRAL can be used to convert the information to the preprocessor internal format. For data formats not recognized by the system, a set of general-purpose data translating subroutines are provided, which will permit the conversion of external data with a minimum of user programming. These utility routines are described in Subsection 3.3.

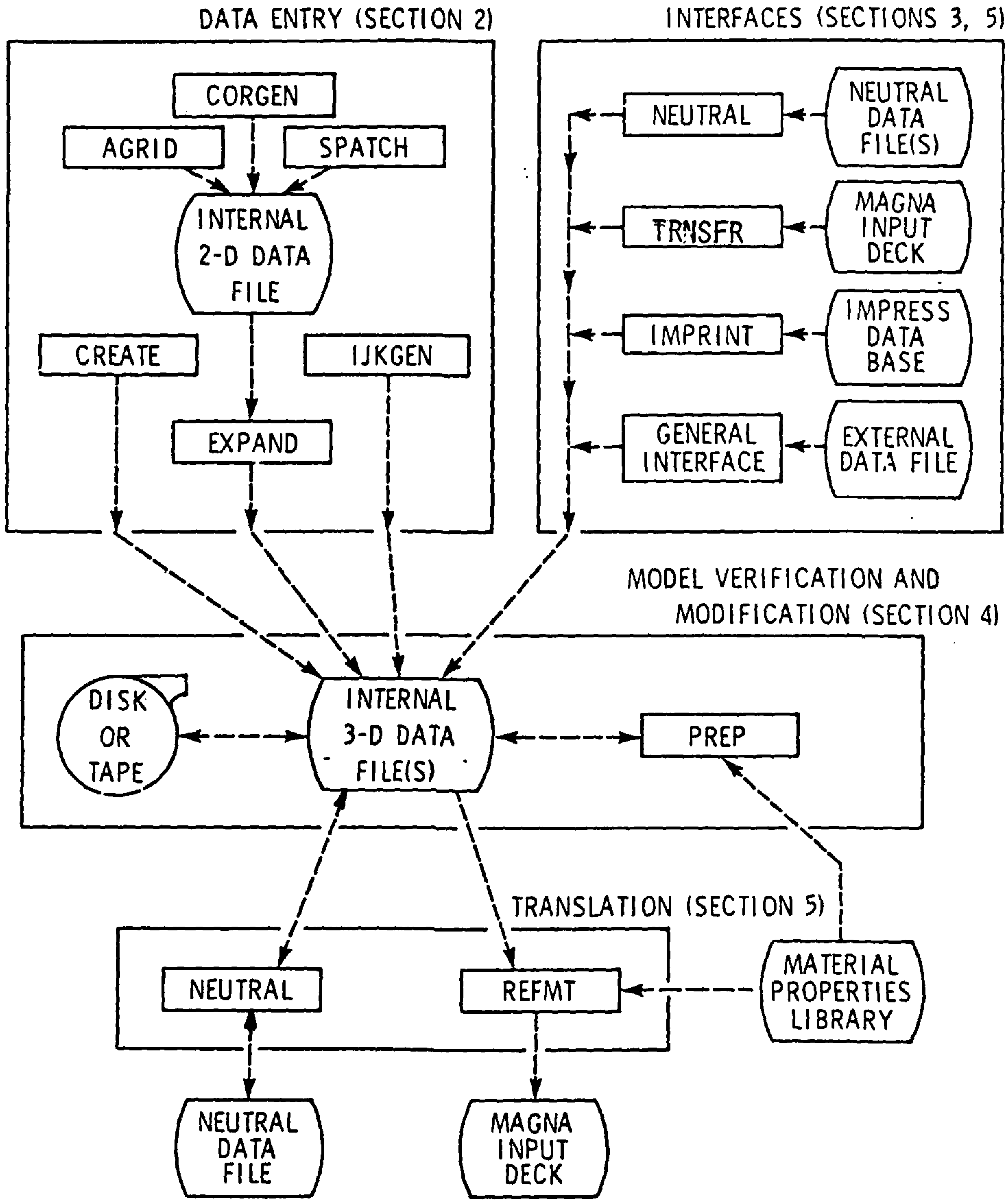


Figure 1.1. Preprocessor Organization.

TABLE 1.1

SUMMARY OF INPUT DATA TYPES FOR DATA ENTRY MODULES

	CREATE	IJKGEN	CORGEN	AGRID	SPATCH
Keyboard	X	X	X		
Data File	X		X	X	X
Digitizer			X		
User Subroutine		X			X

The central preprocessing program, PREP, uses a single file format for data input and output. Each of the data entry and interface utilities creates finite element data files in the PREP standard format; once expressed in this form, the finite element mesh can be reprocessed numerous times using PREP, and may be combined with other models stored in the same form. Data stored in the preprocessor internal format may be retained on auxiliary storage and modified as many times as desired. Printed output and geometry plots are also available at this stage.

Data translators which interpret the output of the preprocessor are of two types: the REFMT processor generates finite element data which is readable directly by the finite element analysis program MAGNA[4], and NEUTRAL produces a formatted data file which may be translated for use with other preprocessors and/or analysis codes. Since NEUTRAL can also be used to translate the neutral file back to PREP format, it is also a useful means of transferring modelling data from one computer system to another. Obviously, the entire output translation step may be replaced if necessary by one's own conversion programs. The PREP data sequence, as it occurs in files output by the PREP processor, is described fully in Section 6 for such applications.

SECTION 2

DATA ENTRY FUNCTIONS

The initial step in generating a model using the present preprocessing system is concerned with the general description of the structural geometry. In this phase, the objective is to create a geometric data base which adequately describes the structure or substructure of interest. The model so defined will be generally too coarse for the actual finite element stress solution, but will be sufficient to identify the structural geometry which may be edited, refined and merged with other model files to result in a final model which will be suitable for analysis.

The data entry modules provided by the preprocessing system are applicable to rather arbitrary structural geometries. The modules discussed in this section detail the provisions for utilizing various forms of model data input to define a 'coarse' finite element mesh. It is anticipated that the user will employ the editing capabilities of the verification and modification modules to refine the coarse mesh to a mesh suitable for analysis.

The CREATE data entry program provides the simplest possible form of input, in which nodal coordinates and element connections are defined explicitly in free format at the keyboard. This mode of input can be useful when large portions of a model can be defined in terms of uncomplicated shapes.

The data generator IJKGEN allows the definition of a model in analytical form, using known equations of a surface or other shape to set up a topologically regular mesh of nodes and elements. Built in generation options, and/or user-written subroutines, may be used in generating the model.

The CORGEN processor assists the user in combining previously defined coordinate data, lofting data existing either in digitizing form or numerical form, and additional data that may be input directly from the keyboard. This program makes use of an intermediate nodal group concept to allow implementation of interpolation schemes to provide for regular mesh generation.

AGRID is a surface-fitting utility program which permits the definition of a regular surface mesh from scattered data which is not suitable for input in any other form. This module uses a special smoothing technique which computes surfaces passing through a number of specified points based on a 'least curvature' criterion.

The SPATCH utility, in conjunction with user-written input routines, facilitates the translation of geometric data stored in the form of bicubic 'surface patches' into the forms recognized by the preprocessor.

Each of these data entry functions is described in further detail in the following Subsections. For modelling data which is incompatible with the data definition utilities discussed below, the reader is referred to Subsection 3.3, which describes a set of general interface routines which can be adapted to the translation of most types of data into the preprocessor formats.

2.1 CREATE - Coarse Mesh Input

The CREATE program is oriented toward problems which involve geometric shapes that might be defined by a very coarse grid of finite elements, even though a much finer mesh would be required for stress analysis. Such a simple model (e.g., a tapered plate defined using a single eight-node brick) can be translated into the preprocessor format using CREATE, and refined for analysis with PREP. CREATE is fully interactive, prompting the user for each of the inputs it requires. There are currently limits of 500 nodes and 100 finite elements.

The access procedures for CREATE are detailed in Appendix C for all computer systems currently supporting CREATE. Once initiated, the program prompts the user with the following pattern of requests:

i.) is this a re-edit session?,
ii.) define a 2-D or 3-D coordinate system to be used,
iii.) input the nodal coordinates,
iv.) edit the nodal coordinates?,
v.) input the element nodal connectivity,
vi.) edit the element connectivity?,
vii.) edit nodal data or element connectivity?

Sample program executions are provided in Section 7.1 for CREATE. A feature is provided which allows for re-editing a previously CREATE'd file to eliminate mistakes. The user is initially asked if the current session is a re-edit session. If the user responds that it is a re-edit session, there must be a file present containing previous output from the CREATE program. Should the user select a re-edit session there will only be one edit pass made through the data.

Data entered to CREATE must be either two-dimensional or three-dimensional in form. For two-dimensional data, two coordinates (X, Y) must be defined for each node; three-dimensional coordinates require a third (Z) value. The user may define nodes that are not connected to the model being defined, but all nodes specified in the element nodal connectivity must be defined. Unused nodes may be eliminated in the PREP module.

Elements are defined in CREATE with reference to the 27-node solid and 9-node planar elements shown in Figures 2.1 and 2.2. If the user has selected a 2-D coordinate system, only four to nine nodes are required to define each planar element. The elements illustrated in Figure 2.2 are typical 2-D elements the user may define. When specifying elements using a 2-D coordinate system, the user must input the element number, the number of nodes to be used to define this element (from four to nine), followed by the nodes themselves. Three dimensional models are defined in the same manner as for 2-D, except that each finite element may contain from four to twenty-seven connected nodes. Since each element may have a different number of nodes, the user must enter the element number, the number of nodes to be used to define the element and the node numbers involved.

The completed model data is written to the default file specified in the access procedures for CREATE. This data file is unformatted and is suitable for input directly to the PREP preprocessing module for modification and reformatting.

2.2 IJKGEN - ANALYTICAL SURFACE GEOMETRY INPUT

The IJKGEN mesh generator is a stand-alone program which creates finite element data from analytical descriptions of the geometry in question. Figure 2.3 demonstrates the types of geometries which lend themselves to the generation scheme used in IJKGEN: in each instance, the shape can be imagined as varying

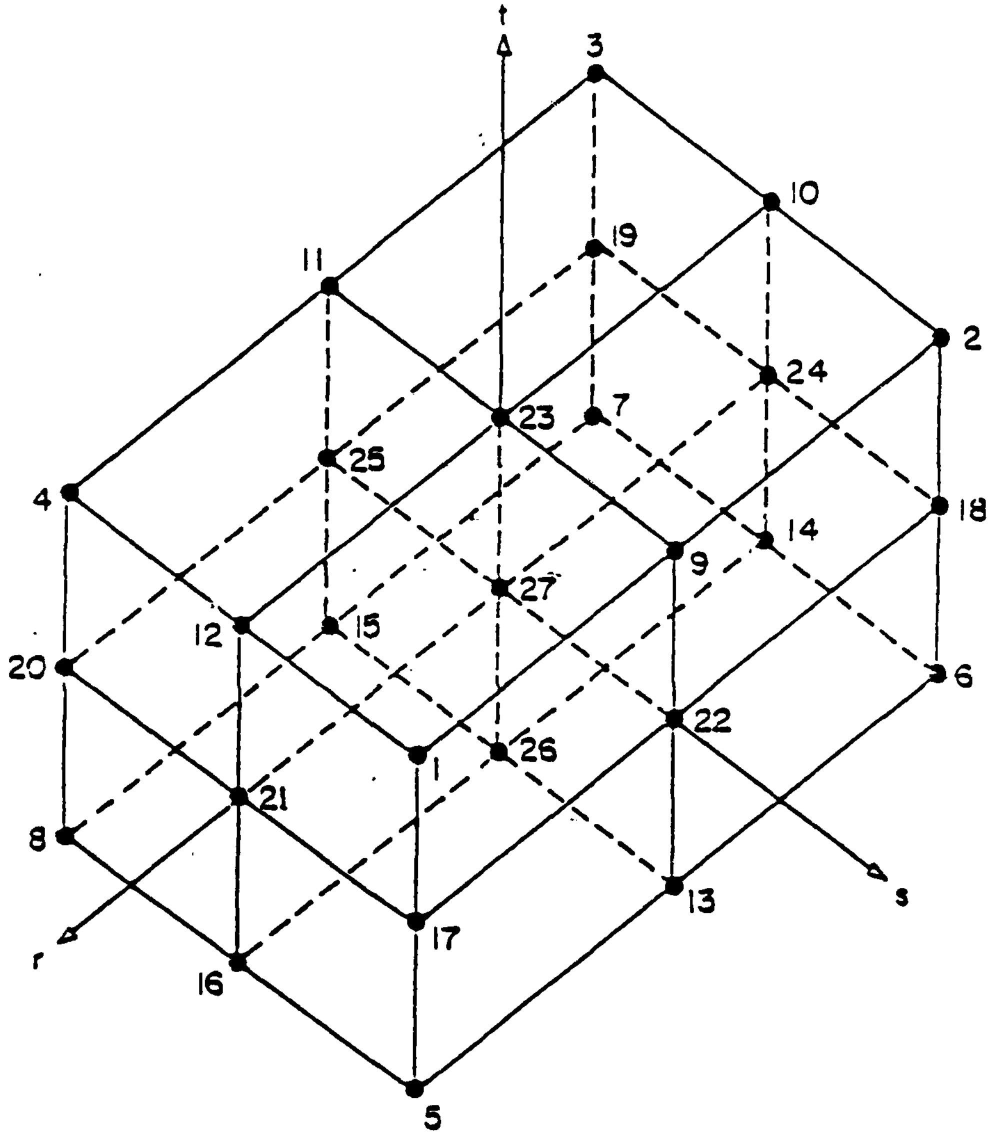
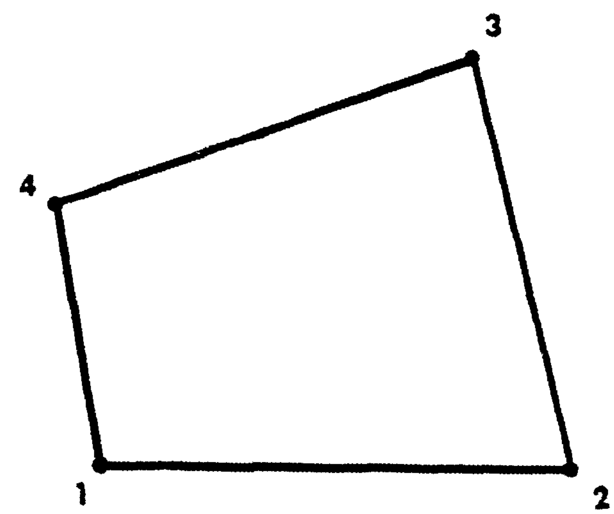


Figure 2.1. 27 Node Solid Element.



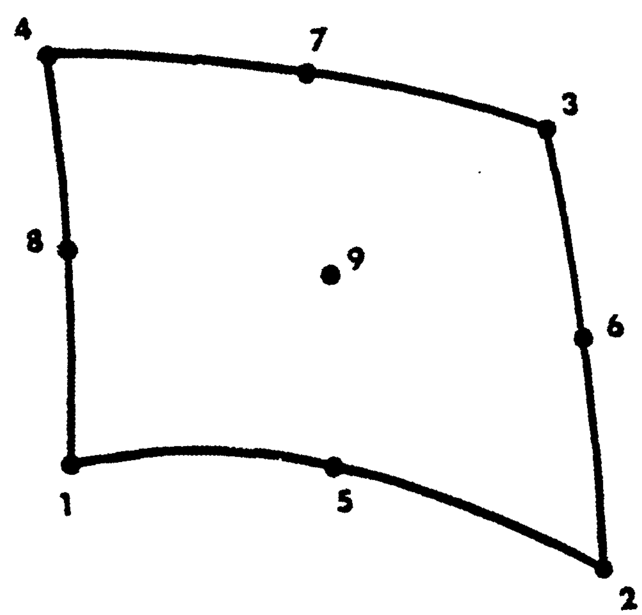


Figure 2.2. Four and Nine Node Surface Elements.

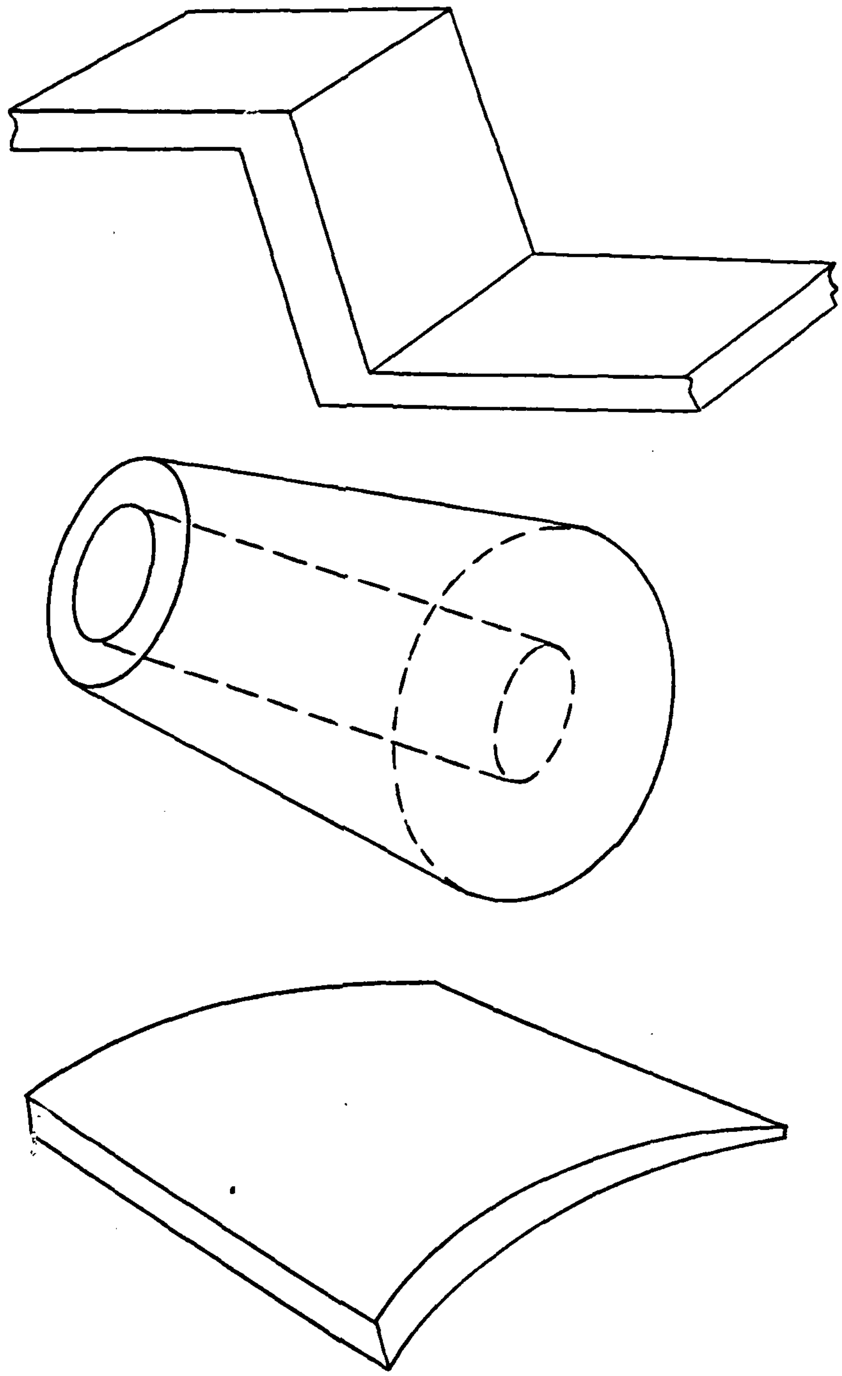


Figure 2.3. Types of Geometries Suitable for IJKGEN.

(at least piecewise) continuously with the "generator indices" I, J and K.

A very simple example of the method of generation in IJKGEN appears in Figure 2.4. Since the three coordinates R, Θ, Z are orthogonal, we might express them in terms of generator indices I, J, K as

$$
\begin{aligned}
R &= R\,(I) \\
\Theta &= \Theta\,(J) \\
Z &= Z\,(K)
\end{aligned}
\tag{2.1}
$$

in which I varies from one to the number of nodes in the radial direction, and so on. Given the limits on R, Θ and Z, IJKGEN would generate a mesh by

- looping over all combinations of (I,J,K),
- for each (I,J,K) combination, "interpolating" for (R,Θ,Z),
- transforming each (R,Θ,Z) to Cartesian (X,Y,Z) coordinates.

Connectivity is generated for the model using the fact that the shape is topologically regular; that is, the connectivity for the cylinder is precisely the same as for a rectangular parallelepiped having the same number of nodes in each direction as the cylinder.

The basic mode of operation of IJKGEN uses a number of built-in options for geometric shapes, including rectangular flat plates, cylindrical shells and spherical shells. Furthermore, two built-in options are provided for interpolating the curvilinear coordinates in terms of I, J and K: (1) uniform mesh spacing; and (2) logarithmic grading, with the ratio of the sizes of the first and last elements in each direction specified by the user. Figures 2.5 and 2.6 illustrate the default options for coordinate systems and mesh spacing in IJKGEN.

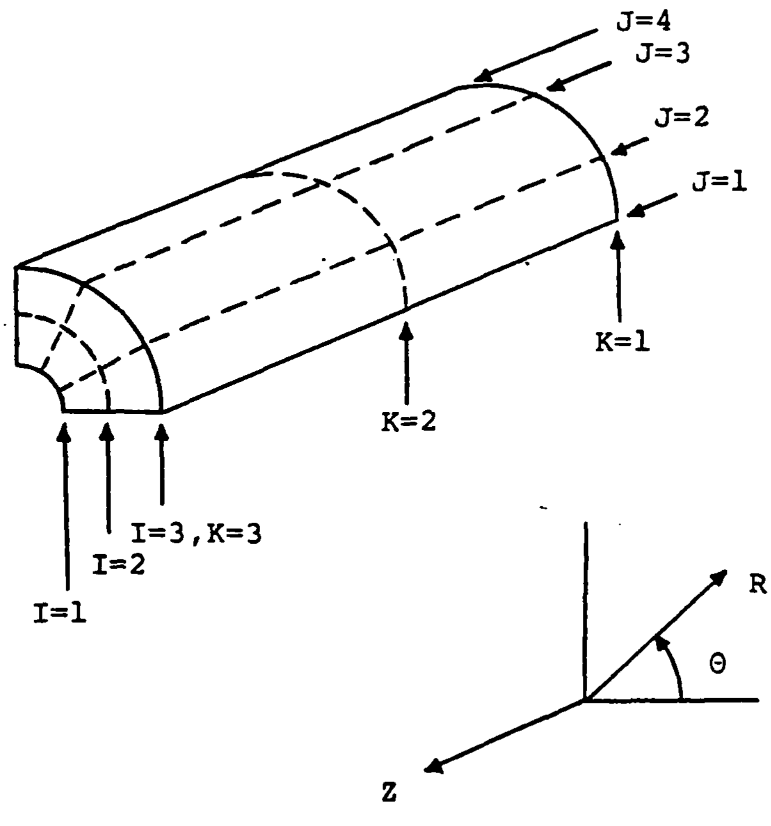


Figure 2.4. Example of IJKGEN Method of Generation.

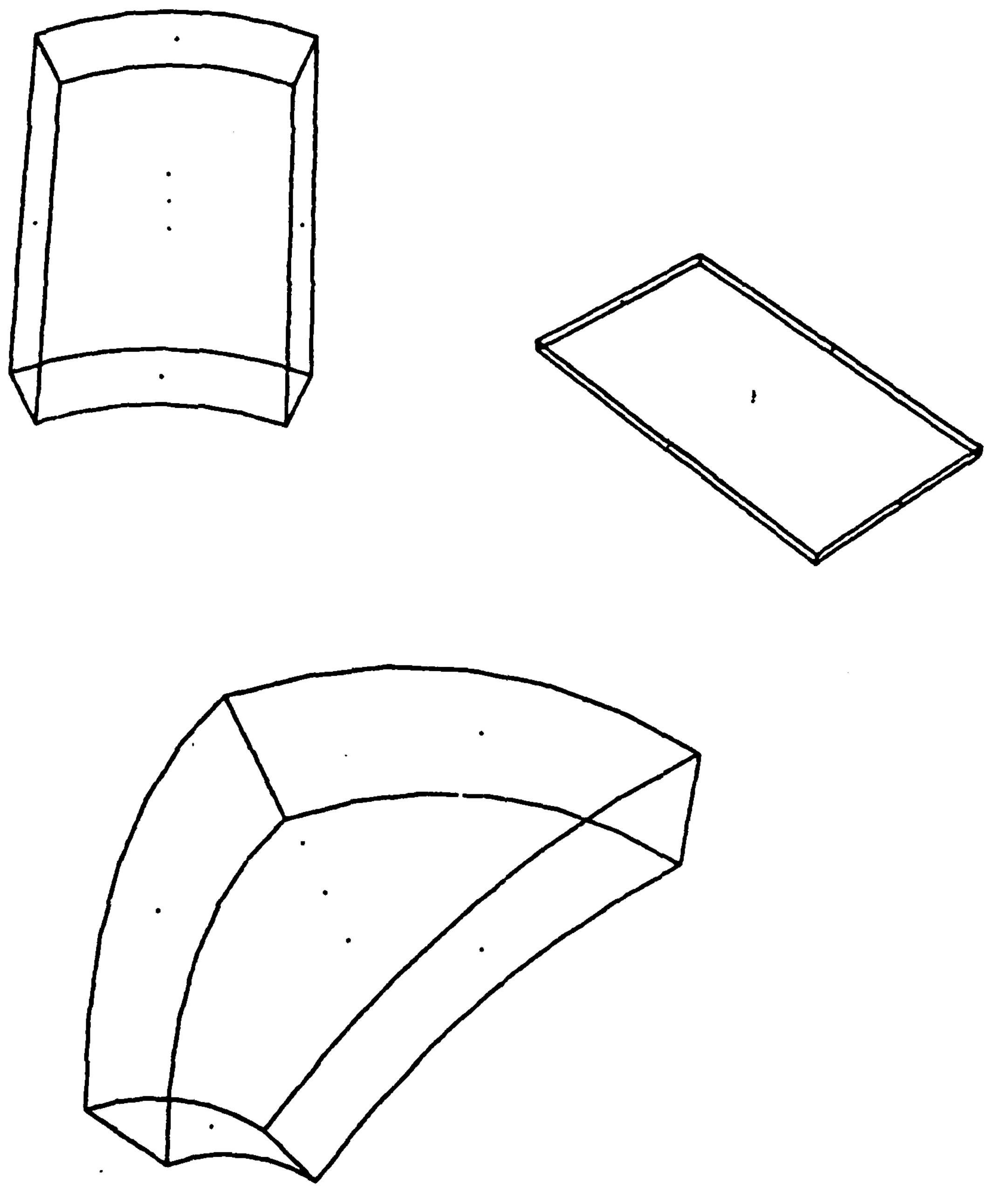


Figure 2.5. IJKGEN Default Coordinate Options.

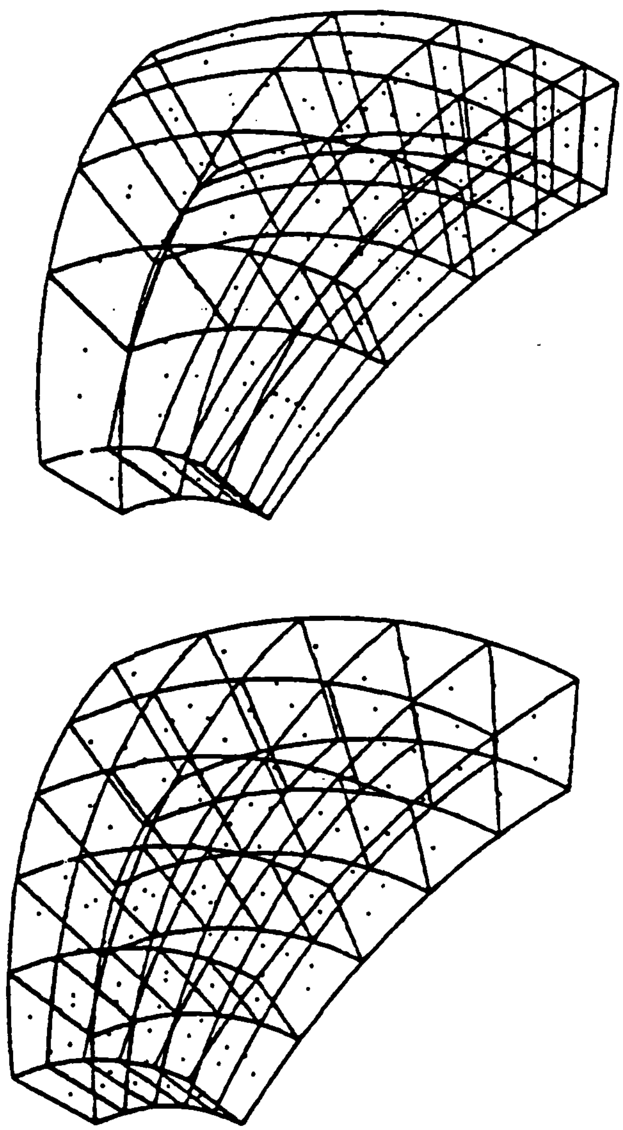


Figure 2.6. IJKGEN Default Mesh Spacing Options.

When the default options in IJKGEN are used, the data required as input includes

- the coordinate system type (Cartesian,cylindrical,spherical),
- the number of elements in each direction,
- the mesh spacing option, and
- limits on the curvilinear coordinate values.

The generated mesh is always composed of 27-node solid elements, which may be reduced to lower-order elements in PREP if necessary (see Section 4.1, the MASK function). To ensure that 27-node elements which are to be MASKed to create 16-node elements (for example) are properly oriented, IJKGEN also allows the selection of a "thickness direction" prior to generating the element connectivity data.

Figure 2.7 shows a sample execution of IJKGEN, using the default options. In this case, the shape is rectangular, the mesh is graded in all three directions, and no thickness direction for the model is specified. Figure 2.8 shows a plot (obtained from PREP) of the finite element model as output by IJKGEN.

Naturally, the simple geometric shapes included in IJKGEN as built-in functions represent the exception, not the rule, for practical applications. When the shape of a model is known in an analytical form other than those included in IJKGEN, three user-written subroutines may be introduced to describe the geometry in equation form:

- SUBROUTINE UINPUT
- SUBROUTINE SURFAC (I,J,K,ALPHA,BETA,ZETA)
- SUBROUTINE CRDTRN (ALPHA,BETA,ZETA,X,Y,Z)

The subroutine UINPUT is called at the beginning of execution, to

```
***********************************************
**********    BEGIN IJKGEN    **********
***********************************************

I J K G E N   -   GENERATION OF GEOMETRIC MESH DATA FOR SOLID,
THICK SHELL OR PLATE FINITE ELEMENT MODELS, USING AN INTEGER -
COORDINATE INDEXING SCHEME.  OPTIONAL USER ROUTINES ARE -
  (1)  SURFAC (I,J,K,ALPHA,BETA,ZETA) - DEFINE MESH GEOMETRY
  (2)  CRDTRN (ALPHA,BETA,ZETA,X,Y,Z) - COORD. TRANSFORMATION
BUILT-IN OPTIONS INCLUDE RECTANGULAR, CYLINDRICAL OR SPHERICAL
COORDINATES, AND UNIFORM OR PROPORTIONALLY GRADED MESH SPACING

  (3)  UINPUT - USER PARAMETER INPUT ROUTINE (INITIALIZE DATA
                IN BLANK COMMON)

.........................................................

**********  USER SUBROUTINE 'SURFAC' NOT GIVEN  **********

BUILT-IN MESH DIVISION OPTIONS ARE AS FOLLOWS -
 (1) - UNIFORM MESH IN EACH DIRECTION
 (2) - GRADED MESH (SPECIFY RATIO OF FIRST/LAST ELEMENT SIZE

ENTER OPTION ( 1 , 2 ) ................2

ENTER THE RATIO OF FIRST / LAST ELEMENT LENGTHS FOR EACH
COORDINATE DIRECTION (ALPHA, BETA, ZETA) (R=1 FOR UNIFORM)

ENTER LENGTH RATIOS (R1,R2,R3) ........3 4 5

**********  USER SUBROUTINE 'CRDTRN' NOT GIVEN  **********

BUILT-IN COORDINATE SYSTEM TRANSFORMATION OPTIONS ARE - - -
(1) RECTANGULAR,  (2) CYLINDRICAL,  (3) SPHERICAL

ENTER COORDINATE SYSTEM OPTION (1,2,3) -1

** PLEASE NOTE THE FOLLOWING CONVENTIONS FOR RECTANGULAR SYSTEM **

 'ALPHA' = X     'BETA' = Y     'ZETA' = Z
A RIGHT-HANDED SYSTEM IS ASSUMED.

ENTER LIMITING SURFACE COORDINATE VALUES -

 1. ALPHA(MIN)    2. ALPHA(MAX)
 3. BETA (MIN)    4. BETA (MAX)
 5. ZETA (MIN)    6. ZETA (MAX)

-5. 5. 0. 13. -2. 1

ENTER THE NUMBER OF ELEMENTS TO BE GEN-
 ERATED IN THE ALPHA, BETA AND ZETA CO-
 ORDINATE DIRECTIONS, RESPECTIVELY .... 3 3 3


 *** BEGIN GENERATION PHASE ***

 NUMBER OF NODES TO BE GENERATED    =  343
 NUMBER OF ELEMENTS TO BE GENERATED=   27

 THE THICKNESS DIRECTION OF THE MODEL (IF ONE EXISTS, AS IN A
 THICK SHELL) MUST BE IDENTIFIED TO ORIENT ELEMENTS PROPERLY.

 OPTIONS ARE (1)ALPHA, (2)BETA, (3)ZETA, OR (4)(UNIMPORTANT).

 ENTER THICKNESS DIRECTION CODE (1,2,3,4) -4

 *** DATA GENERATION COMPLETE ***

 ****** IJKGEN  TERMINATED ******
     STOP
     025500 MAXIMUM EXECUTION FL.
       .147 CP SECONDS EXECUTION TIME.
COMMAND-
```

Figure 2.7. Sample Execution of IJKGEN.

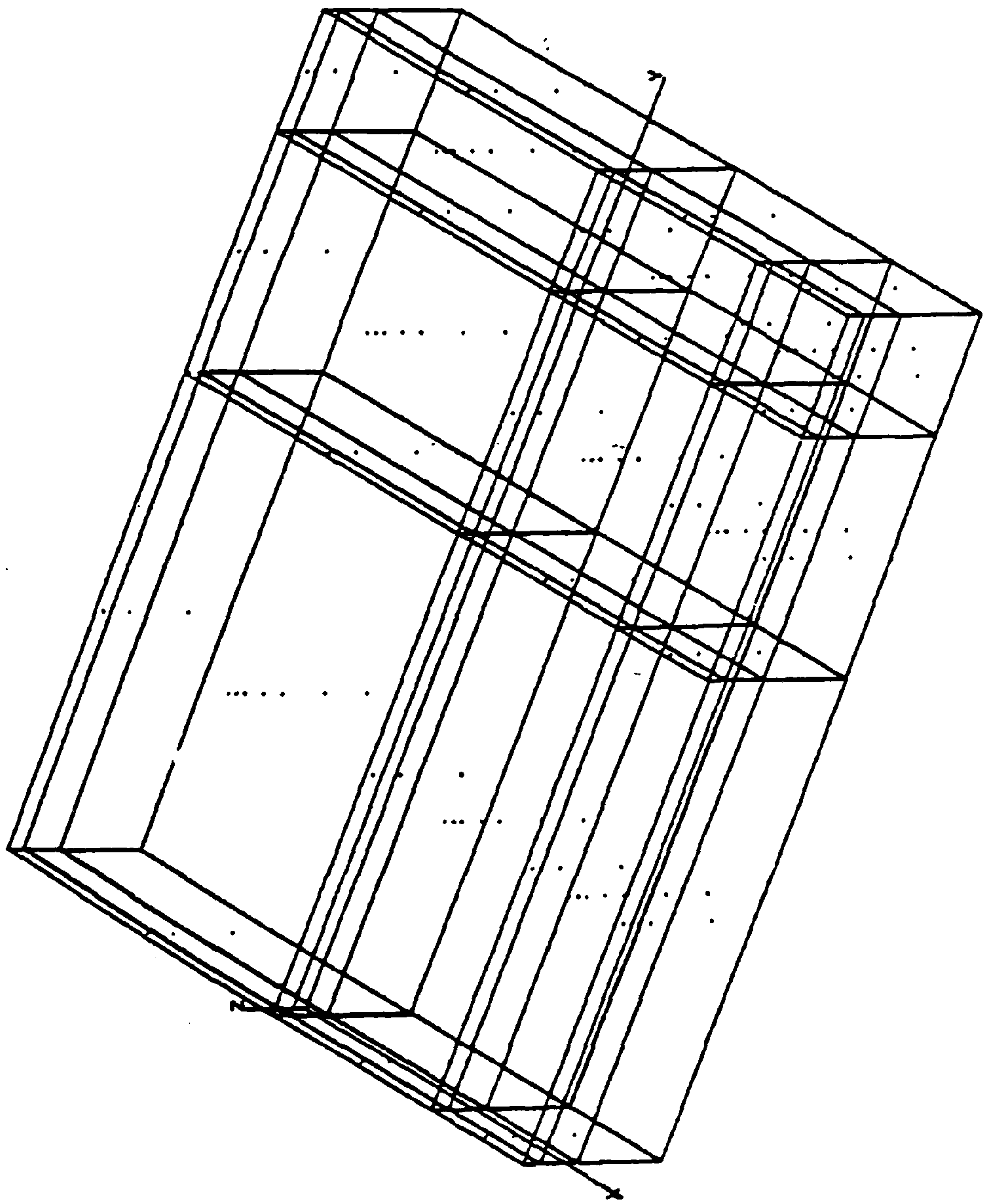


Figure 2.8. Plot of IJKGEN Sample Model from PREP.

allow initialization of parameters (e.g., in COMMON blocks) which might be needed in the other two routines. Subroutine SURFAC computes the values of the curvilinear coordinates (ALPHA,BETA, ZETA) as functions of (I,J,K), and therefore allows the introduction of non-orthogonal coordinates and variable mesh spacing. The CRDTRN routine accepts curvilinear coordinate values (ALPHA, BETA,ZETA) and returns Cartesian coordinates X, Y, Z; in effect, CRDTRN defines the coordinate system in which IJKGEN generates the finite element mesh.

Figure 2.9 shows a model of the B-1 bomber left windshield generated with IJKGEN [6]. In this case, UINPUT has been used to read in "key points" for the windshield geometry, which is cylindrical with irregular boundaries; a two-element mesh which uses only the key points and a few intermediate points is shown in Figure 2.10. SURFAC is used to generate intermediate points on the windshield by linear interpolation, and CRDTRN performs the final transformation to Cartesian coordinates.

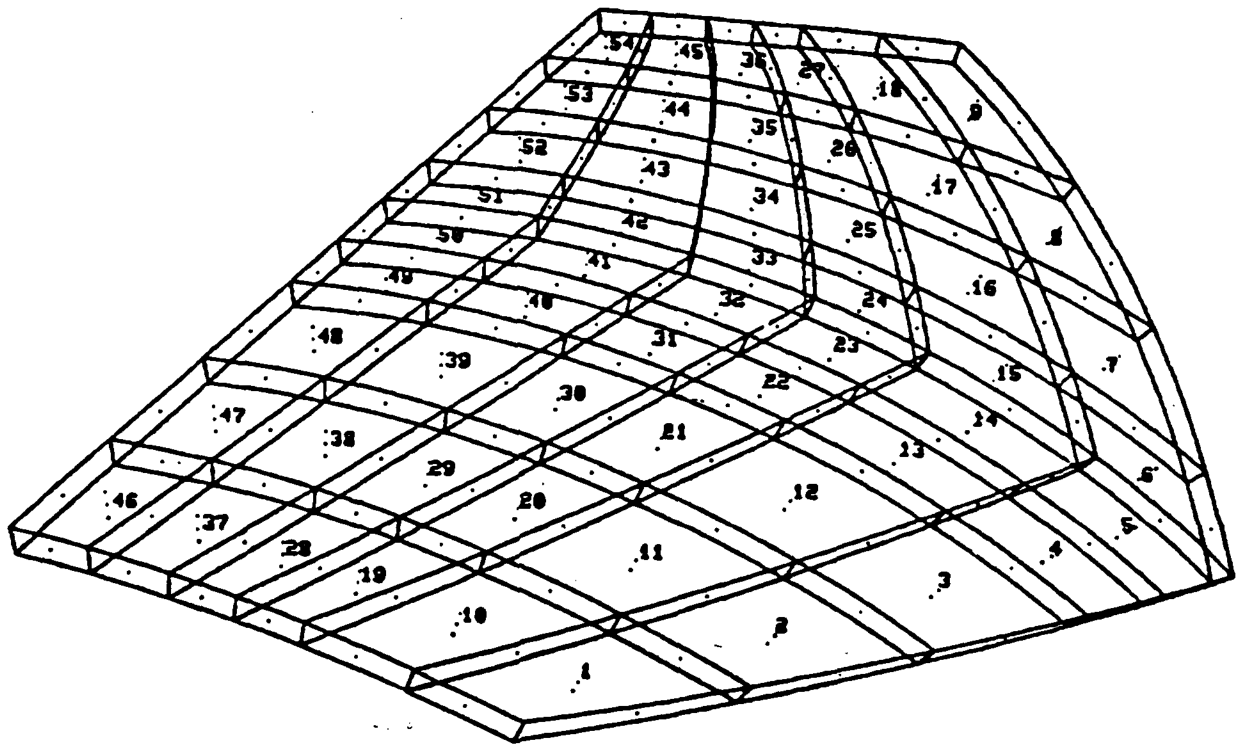


Figure 2.9. Portion of B-1 Bomber Windshield.

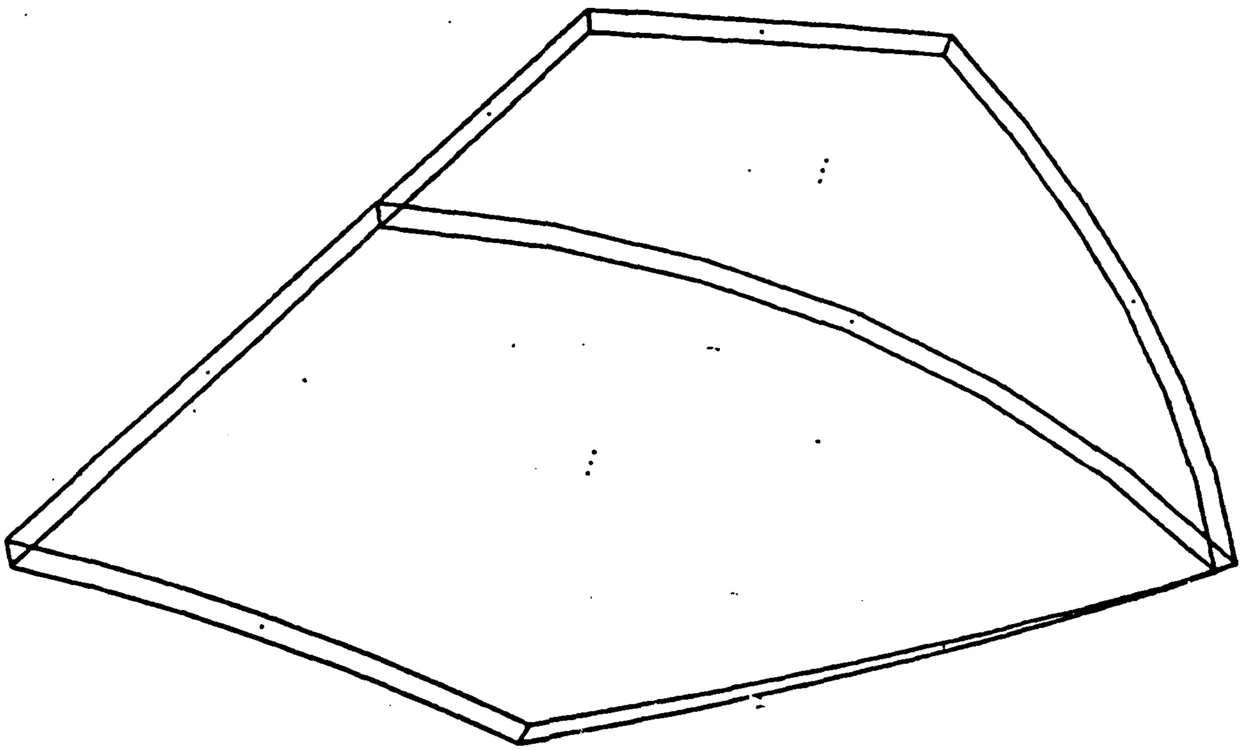


Figure 2.10. Coarse B-1 Windshield Mesh using Key Point Coordinates.

2.3 CORGEN - General Surface Geometry Input

CORGEN is a general purpose finite element preprocessor program designed to interpret surface lofting type data, either from sets of coordinate data or from digitized data in the form of a geometric description. The program will also accept direct creation of model geometry and element connectivity from keyboard input. Geometric data input may come from a variety of media such as disc files, keyboard or digitizer tablet, or a combination of these. Reference points entered by any of these means may be used to generate data along lines, circular arcs or more general curved paths.

Data entry is performed according to "groups" of points where data from an individual lofting plane (cross-section) will correspond to a single group as illustrated in Figure 2.11. For each group of data input, the user may select a different input medium (e.g file, keyboard or digitizer), a separate coordinate system transformation (Cartesian, cylindrical, or spherical), independent coordinate translations and other parameters which define the placement of the group within the global coordinate system. Input from the digitizing tablet, for instance, is always given in planar form (2-D) but may be arbitrarily located in three dimensional space by specifying the location of a positional digitizing plane using any three non-collinear points lying in the plane.

Ultimately, the geometric data presented to CORGEN must be arranged in nodal groups containing similar numbers of points. Data which does not conform to this scheme may be interpolated or otherwise modified after being input to CORGEN. The commands available in CORGEN are generally of three types: (1) data input commands; (2) generation/interpolation commands; and (3) element generation commands. Input commands define an input medium, or cause data to be read from an already-defined medium. Generation and interpolation commands initiate internal generation of nodes,

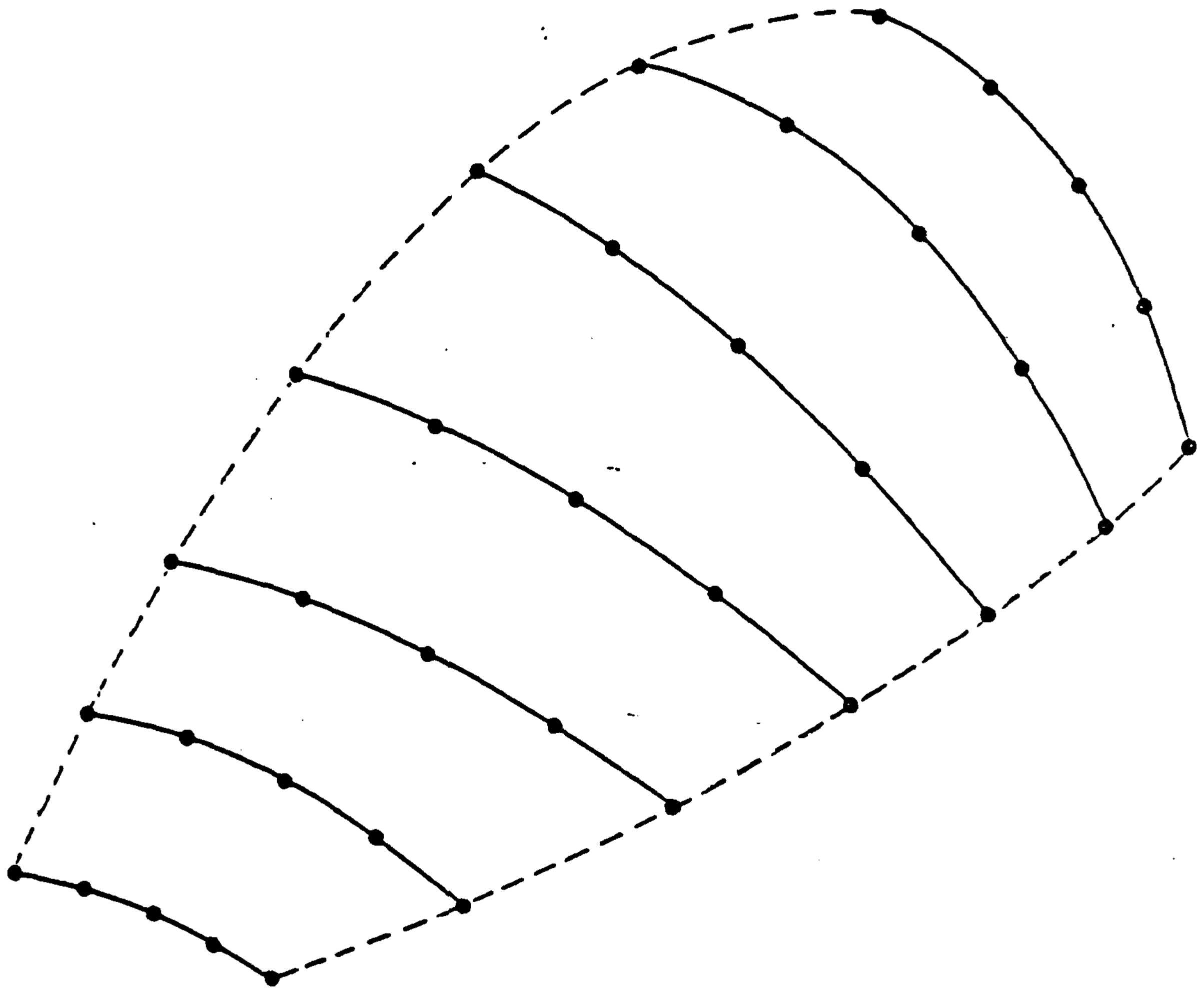


Figure 2.11. Lofting Surface.

usually for the purpose of "filling out" one or more nodal groups. Element generation commands cause the surface model to be completed by defining surface element connection data.

Often the available data is not suitable for generating a finite element mesh directly, due to poor point spacing or density. In such situations, interpolation of point data may be performed in CORGEN, using the method described in Reference [5]. Interpolation of new nodes between existing nodes, or placement of entire new node groups between previously-defined cross-sections produces smoothly interpolated data, even in situations where spline fitting may be inadequate. Figure 2.12 illustrates the salient differences between these methods.

Whenever the user specifies the input of nodal coordinate data, prompts will request necessary information for forming groups from that data. For example, if the user were to specify file input of nodes, the user would be asked to specify the number of nodes per group to be input as well as the number of groups that should be input concurrently containing that number of nodes per group, before returning control to the user. Simple modelling requirements will entail no special interaction by the user to alter or select groups, since CORGEN is structured to assume the user has entered nodes and groups in the order they are to be utilized in element generation.

Element generation in CORGEN defaults to utilize all groups currently defined, in the order they were defined. This may be altered by the user in special cases where new groups are created or groups were not input in the correct order for the generator. Additionally, the user should be aware that when a group is specified for interpolation, the new group will replace the old group. This can result in significant disarray of node numbering sequences in the completed model. Provisions are made for renumbering the nodes in the PREP module to eliminate problems with non-optimal node numbering. Should the user create groups that

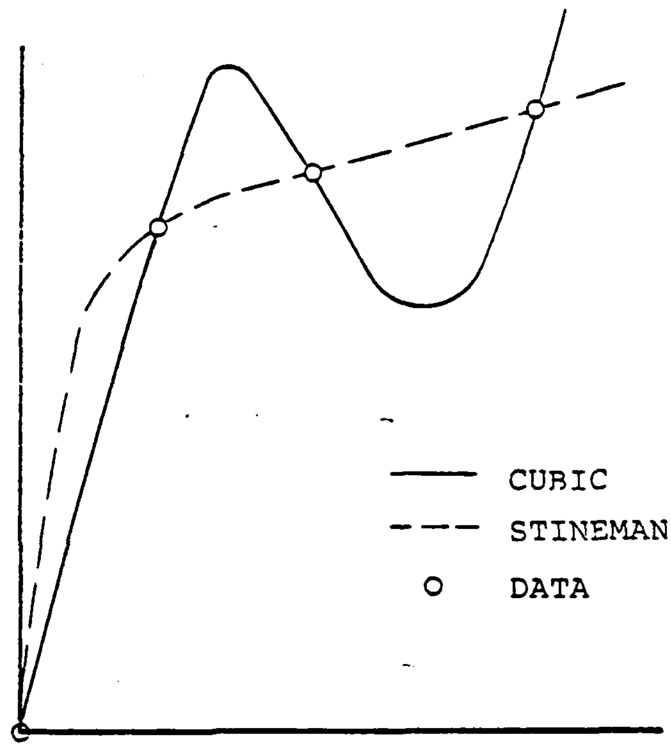


Figure 2.12. Stineman Interpolation.

are not built with equivalent numbers of nodes, CORGEN will automatically invoke the interpolation routines to homogenize all groups specified for element generation. Element generation has two options available for positioning midside nodes. The program default is to always generate interpolated nodes between existing nodes such that midside nodes are properly located.

Occasionally, it is necessary to extensively alter input geometry data to achieve an appropriate model for analysis. Provisions are available for user creation of new groups by specifying nodes to be used for interpolation and by user selection of groups to be utilized in element generation. Additionally, a number of options exist for altering node, group and element data to meet a variety of modelling requirements. In the event the user wishes to utilize the element generator but does not wish to have new nodes interpolated then the groups must be defined such that an odd number of nodes are located in each group and that an odd number of groups are selected for element generation.

2.3.1 Interactive Program Execution

The interactive capabilities of CORGEN are summarized in Table 2.1. The program is essentially command driven, but oriented such that the user is displayed only a menu of those options or commands immediately relevant. Extensive help facilities assist the user in understanding what options are available and provides information relevant to the current stage of model development.

Initial input to CORGEN consists of establishing the proper coordinate system transformations, implementing any coordinate translations, resetting default nodal attributes, and any other parameters that affect the input of nodal coordinate data. Figure 2.13 lists the default options of CORGEN and what commands are used to alter them. Once the user has adequately defined

TABLE 2.1

CORGEN INTERACTIVE PROCESSING CAPABILITIES

	Node	Group	Element
Input			
File	X		X
Keyboard	X	X	X
Digitizer	X		
Editing			
File			
Keyboard	X	X	X
Digitizer	X		
Interpolation	X	X	
Auto-generation		X	X
Auto-interpolation	X		
Generation (other)	X		
Save/Restore	X	X	X

Other features:

- o 2-D lofting plane specification in arbitrary space
- o Default values for thickness and coordinates
- o Coordinate transformations and translations

Default Options for CORGEN Program Execution

Function	Default	Command
Auto Interpolation	Yes	INTErpolate
Auto Mesh Generation	Yes	ELEMent
Coordinate System	Cartesian	COORdinate
Digitizer Pick Tolerance	5% horizontal axis	TOLErance
Lofting Plane	(Ø,Ø,1) X vertical screen Y horizontal screen Z into screen	PLANe
Translation Factors	x=Ø, y=Ø, z=Ø	COORdinate
Thickness	1.Ø	THICk

Figure 2.13. CORGEN Default Options and Commands

Menu Paths

A. File input of nodal coordinates, automatic interpolation, and mesh generation:

NODE, FILE, READ, FINI, STOP.

B. Keyboard input of coordinates, order groups for interpolation then selective element generation:

NODE, KEYB, READ;
GROUP, ALTER;
ELEM, GENER;
STOP.

Figure 2.14. CORGEN Command/Menu Paths.

the input coordinate system, a mechanism should be selected for nodal coordinate input. In defining the nodes to CORGEN, the user must be cognizant of the need to incorporate nodes into 'nodal groups'. All nodal input sequences will question the user as to how many nodes are to be placed in the next input group. There should generally be an equal number of nodes in all groups or the interpolation functions will automatically be invoked to reconfigure the groups so that all are equal.

The menu paths of CORGEN are detailed in Figure 2.14. The user is encouraged to follow a straightforward approach to model generation by first defining nodes in nodal groups, then selecting any needed interpolation and finally generating elements. Many of the features in CORGEN are automatic and will be invisible to the user if nodal groups are defined properly at the outset. Each master command will be discussed below. All commands are enabled by typing the one to four letter abbreviation enclosed in parentheses in the commands listing. The command listings are always available by typing COMMands. Master commands are those commands which are available for the COMMAND..: prompt.

2.3.2 COMMANDS LISTING

The user has available a number of command options at all times. The program is structured to display only those commands which are immediately neccessary to what the user is doing based on previous commands entered. The master commands are listed in Figure 2.15. The master commands will provide the user with more specific options to accomplish particular tasks by requesting more specific input or data. The COMMands command will always display the current optional commands available to the user. Master commands are always accessible by the user, unless specific numeric data has been requested such as nodal coordinate values. The program default is to provide a listing of the available commands only when the user types COMMands.

CORGEN MASTER COMMANDS

COMMANDS..:

```
(COMM)ANDS LISTING
(HELP)
(NODE) DATA
(ELEM)ENT DATA
(GROU)P DATA
(SAVE) MODEL
(REST)ORE MODEL
(COOR)DINATE SYSTEM
(INTE)RPOLATE DATA
(DISP)LAY VALUES
(FINI)SH MODEL
(THIC)KNESS
(STOP)
```

Figure 2.15. Master Commands for CORGEN.

2.3.3 COORDINATE SYSTEM

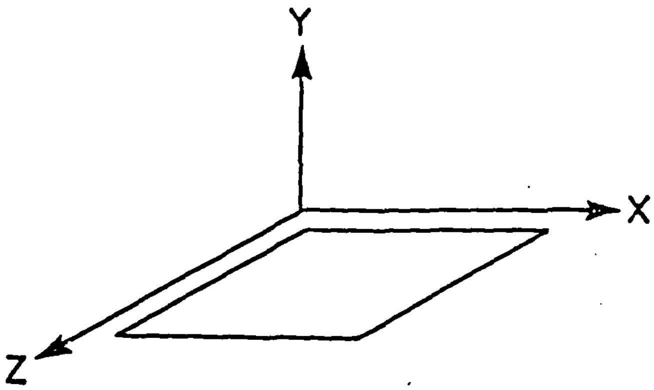
Three automatic coordinate transformations are built into CORGEN. The Cartesian (rectilinear) coordinate system is default. Spherical coordinate conversions may be specified for data input as (R,PHI,THETA) where R is the radius of a sphere and THETA and PHI represent angles in degrees, locating the points on a spherical surface. Cylindrical coordinates conversion is also available for model data defined by (R,THETA,Z) coordinates, where R is the radius from the center of a cylinder, THETA is the angle in degrees of the point about the circular axis and Z defines the longitudinal distance along the cylinder axis. The three coordinate systems are illustrated in Figure 2.16. The COORdinate command selects the current coordinate system until a new one is specified.

The user is queried if coordinate translations of data from the origin are desired whenever the COORdinate command is executed. The translation values will remain in effect until changed or disabled by again executing the COORdinate command. Coordinate translations are specified as positive or negative magnitudes to direct the node in the X, Y and Z directions respectively. The translations are applied to input coordinate data after the coordinate transformations.

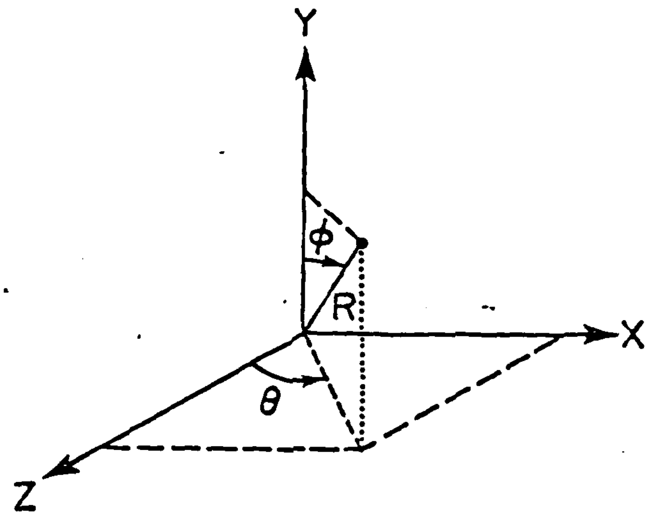
2.3.4 DIGITIZING DATA COMMAND

DIGItize is an input command designed for digitizing nodes and element connectivity utilizing lofting planes oriented arbitrarily in three dimensional space. Data may be input to CORGEN in the form of digitized points from sources such as lofting drawings.

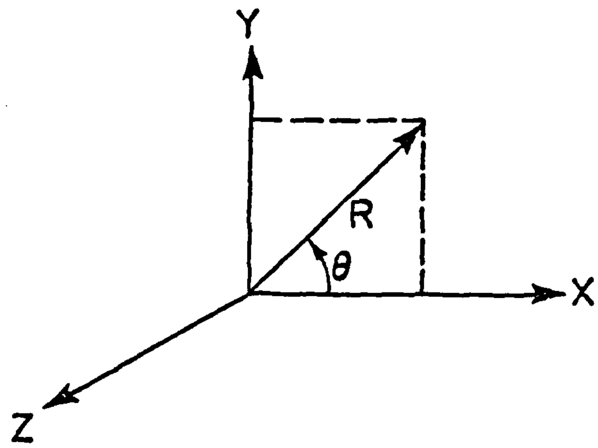
The DIGItize command is currenty implemented to operate



a. Right-handed Cartesian Coordinate System Definition (X, Y, Z)



b. Right-handed Spherical Coordinate System Definition (R, ϕ, θ)



c. Right-handed Cylindrical Coordinate System Definition (R, θ, Z)

CORGEN COORDINATE SYSTEMS

Figure 2.16. Right Handed Cartesian (X,Y,Z) Coordinate System;
Right Handed Spherical (R,ϕ,θ) Coordinate System;
Right Handed Cylindrical (R,θ,Z) Coordinate System.

with Tektronix graphics tablets, terminal (cross-hair) screen cursors, or terminals that emulate Tektronix graphics terminals. The command uses both command prompt and question/answer techniques to establish the digitizing area. The user will be prompted initially for input to identify the type of graphics terminal being used; the type of digitizing to be done (screen cursor or tablet); the location of a menu area if one is desired; the definition of a lofting plane; and the orientation of the drawing to be digitized on the tablet. Once these initial items are defined the user is required to begin digitizing coordinate points. After the user has completed input of points for the first lofting plane, a menu of command options is displayed to allow the user to redefine the lofting plane; add, move or delete nodes; create or edit elements; or to reset default values of parameters such as layer thicknesses or coordinate translations. Figure 2.17 illustrates the initialization information required.

When the user selects the tablet input option, the program must request information to locate the model on the tablet and to set a user coordinate system independent from that on the tablet. In this case the user must first digitize the point that will represent the origin (0,0) of the first drawing to be input. This point is followed by a second digitized point which should define the maximum point along the USER'S horizontal axis. The user should note that the user's horizontal and vertical axes do not need to align with the tablet (or terminal) horizontal axes as the program will provide the appropriate coordinate translations.

An optional 'menu' area is allowed to facilitate program execution. This menu consists of 16 blocks arranged in a 4 by 4 matrix as illustrated in Figure 2.18. When the menu area option is selected it allows the user to quickly exit the digitizing mode or to easily undo any operation the user may have not really wanted to perform. In addition, the menu area can be used to designate any one of up to nine different predefined thickness

```
PLOTTING PACKAGES:
   1 - DI-3000
2 - TEK PLOT10
3 - NO PACKAGE
ENTER PACKAGE TYPE (1-3)....: 2
TERMINAL TYPES:
   1 - NONGRAPHICS
   2 - TEK 4006
   3 - TEK 4010, 4012, 4052
   4 - TEK 4014, 4016
   5 - TEK 4014 EG
   6 - TEK 4114
ENTER TERMINAL TYPE...: 5
DIGITIZER TYPES:
   1 - TERMINAL W/CURSOR
   2 - TERMINAL W/TABLET
   3 - TABLET STAND-ALONE
ENTER DIGITIZER TYPE (1-3)....: 2
```

```
ESTABLISH A TABLET MENU AREA FOR
CONTROLLING DIGITIZING? (Y,N)..: Y
  THE USER MUST DIGITIZE THE LOWER
  LEFT AND UPPER RIGHT CORNERS OF
  THE TABLET MENU AREA, THEN ENTER
  THE NUMBER OF HORIZONTAL AND
  VERTICAL MENU OPTIONS AVAILABLE.
  DIGITIZE THE LOWER LEFT MENU AREA..:
  DIGITIZE THE UPPER RIGHT MENU AREA..:
```

```
TABLET - SET USER ORIGIN AND AXES
TO ORIENT THE MESH AXES ON THE
TABLET. THE USER MUST DIGITIZE
TWO POINTS - THE HORIZONTAL
ORIGIN OF THE MODEL, THEN THE
MAXIMUM HORIZONTAL VALUE ON THE
HORIZONTAL AXIS. THE USER-ASSIGNED
COORDINATES FOR THESE TWO POINTS
MUST THEN BE ENTERED TO PROPERLY
SCALE THE DIGITIZED DATA. DIGITIZE
THE ORIGIN OF THE HORIZONTAL AXIS.:
ENTER THE COORDINATES FOR THIS POINT(X,Y).:  0.,0.
DIGITIZE THE MAXIMUM HORIZONTAL POINT:
ENTER THE COORDINATES FOR THIS POINT(X,Y).:  100.,0.
```

Figure 2.17. DIGItize Command Initialization Questions.

```
SCALE IS:  1 USER UNIT =   65.300 UNITS.

THE CURRENT LOFT PLANE IS DEFINED
BY THE FOLLOWING VECTORS.:
  V1 = (     1.0000,     0.0000,     0.0000)
  V2 = (     0.0000,     0.0000,     0.0000)
  V3 = (     0.0000,     1.0000,     0.0000)
  CHANGE THIS LOFT PLANE? (Y,N)....: N

DO YOU WISH TO INPUT THICKNESS
VALUES?(Y,N)....: Y

WHICH THICKNESS OPTION?
  1 - INPUT A THICKNESS VALUE FOR EACH POINT
  2 - SET ONE THICKNESS FOR ALL POINTS
  3 - DEFINE LAYERED THICKNESS INDICES
      ASSOCIATED WITH EACH POINT OR GROUP
ENTER THICKNESS OPTION (1,3)....: 2

ENTER THE OVERALL THICKNESS (>0)....: 0.75

THE PICK TOLERANCE =    4.00
DO YOU WANT TO CHANGE IT? (Y,N).: N
  USER MUST DIGITIZE NODES BEFORE ANY
  OTHER COMMAND MAY BE EXECUTED.
  DIGITIZE NODES. WHEN FINISHED
  DIGITIZE THE 'R' CHARACTER IN
  THE MENU BOX, OR DIGITIZE THE
  SAME POINT 3 TIMES.  DIGITIZE
  THE 'D' TO DELETE THE LAST NODE
  INPUT.

  DISPLAYED POINTS ARE THOSE
  ALREADY PLOTTED BY USER.
  BEGIN DIGITIZING POINTS:
  READY? --> Y
```

Figure 2.17. (Continued)

7	8	9	D
4	5	6	E
1	2	3	R
0	0	.	R

Figure 2.18. DIGItize Command Menu Block.

values to each node as it is digitized, in lieu of predefining only a single thickness value for all nodes input. The 16 blocks are defined to be:

COMMAND	BLOCK LABEL 1-9	0	R	E	D	.
DIGNOD	thickness index (predefined)	none	return	return	delete last node	none
DIGELE	none	none	return	return	delete last element	none
MOVNOD	not available - user must digitize the same point three times to exit this command.					
DELNOD	none	none	return	return	restore last node	none
DELELM	none	none	return	return	restore last element	none
LOFT	-- not used --					

Note that the "0" and "R" blocks appear twice each in the menu area, as shown in Figure 2.18. The menu area may be located anywhere within the available digitizing area, and may be set to any size which is convenient for the user. The specified location of the menu area is given by first digitizing the lower left corner of the menu block, then digitizing the upper right corner of the menu block. The program will display the menu block when it enters the DIGMOD command. Each time the DSPLAY command is executed the menu area will be redrawn. To select any item in the menu area the user need only place the digitizing stylus within the block surrounding the character desired.

A default lofting plane is defined which places the global (3-D) X axis along the local (2-D) Y axis (vertical screen); the global Y axis lies along the local X axis (horizontal screen or tablet); and the global Z axis is positive into the terminal screen (away from the user). This is illustrated in Figure 2.19. The user may alter the lofting plane by specifying any three noncollinear points that lie within the desired plane. The LOFT command contains all further discussion of this option.

The user is given the option of specifying thickness values to be associated with each point input. Three options exist for specifying the thickness values. First, the user may input a thickness value for each point AS THEY ARE BEING DIGITIZED. The program will prompt the user for each coordinate thickness value after it has been digitized. A second option allows the user to specify a single thickness value to be assigned to all nodes as they are digitized. This single thickness value will remain in effect until changed by the user with the THICK command. The third option allows the user to define up to nine different thickness values. These values are indexed in the order they are input to the program (e.g. first value = index 1) and are referenced by their index value. Once a point is input by the user, an index (menu area numbers 1-9) must then be digitized to define the thickness at that node.

To complete the initialization, the program displays the current pick tolerance (resolution) in user units and queries

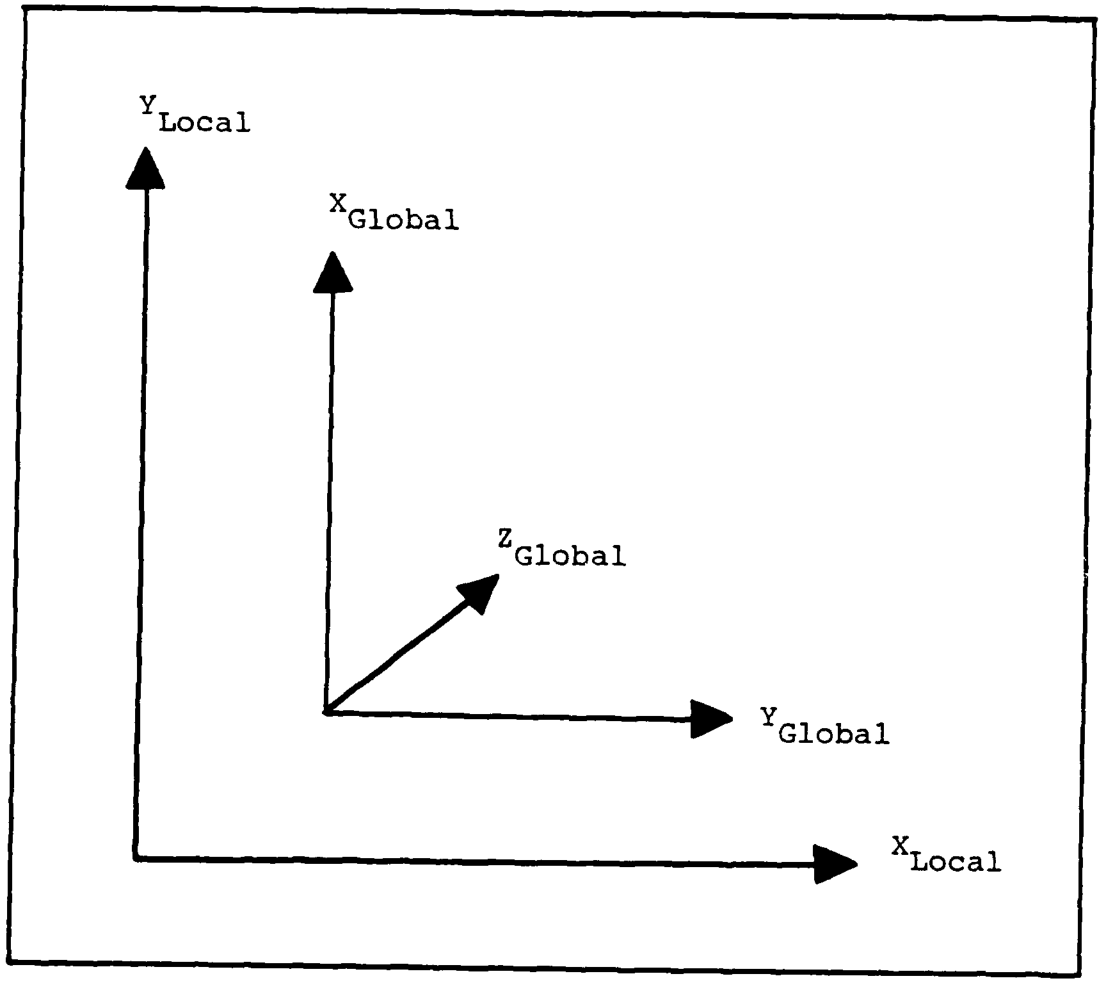


Figure 2.19. CORGEN Default Lofting Plane.

the user if a change is desired. A smaller pick tolerance will require the user to be more exact in selecting nodes or elements for editing or deleting. It is recommended that the user select a smaller tolerance if the coordinates to be input lie relatively close to each other (i.e. within 5% total distance).

Once the initialization is completed the program will automatically execute the DIGNOD command. The DIGNOD command may be exited at any time into the command mode. Figure 2.20 lists the various commands available in DIGItize and a list of the HELP command output. Each command is discussed below.

DIGNOD allows the user to specify the input of points. The user also must define a thickness value for each point input. As discussed above there are several options that allow the user to select a constant thickness value for all points; to specify via the keyboard a different thickness for each point input; or a means of specifying two to nine thickness and digitizing the index of one of those to associate with the current point. The user must have selected the menu option in order to use the indexed thickness values options. Once the user has defined enough nodes he may exit by digitizing the same point three times in succession or by picking the 'R' or 'E' blocks in the menu area. If, at any time, the user wishes to delete the last node entered, simply digitize the 'D' block in the menu area. Repeated use of the 'D' block will continue to delete nodes from the highest numbered node down.

The DIGELE command allows the user to specify element connectivity either by keyboard input or by picking currently defined nodes with the digitizer. At least four nodes must be specified to create an element and as many as nine nodes may be given. Should the user desire to utilize the digitizer to delete elements, then at least five nodes must be given where the fifth node corresponds to local node number 9. Figure 2.21 illustrates the order nodes are to be given to define an element

```
ENTER COMMAND: HELP

CONTROL COMMANDS

======= ========

COMMAND          FUNCTION

------           ---------

DIGNOD  -  DIGITIZE NODES (REQUIRED FIRST)

DIGELE  -  DIGITIZE ELEMENT CONNECTIVITY

DELNOD  -  DELETE A NODE

DELELE  -  DELETE AN ELEMENT

DSPLAY  -  DISPLAY ALL DIGITIZED POINTS

HELP    -  PRINTS THIS LIST

MENU    -  CHANGE MENU LOCATION

MOVNOD  -  MOVE A NODE TO A NEW POSITION

LOFT    -  DEFINE A NEW LOFTING PLANE

SAVE    -  SAVE MODEL DATA

STOP    -  STOP PROGRAM
```

Figure 2.20. DIGItize Command HELP Feature.

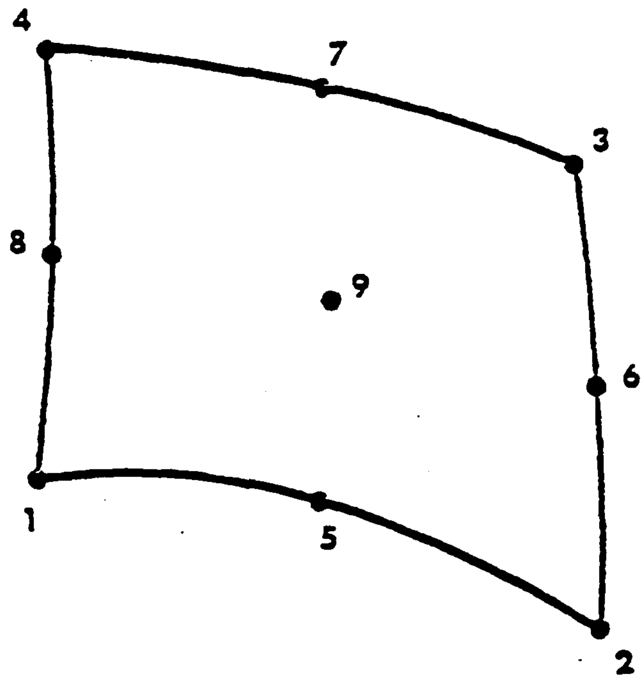


Figure 2.21. Biquadratic Surface Element Generated in CORGEN.

either by keyboard or by digitizer. All four corner nodes are required to define each element, while the remaining five nodes are optional. The user must specify the highest local node number to be defined as the total number of nodes to be given for an element connectivity. For example, if the user wishes to give 5 node numbers to define an element, but have them correspond to local node locations 1-4 and 9, the connectivity list will be 1,2,3,4,0,0,0,0,5 where all intermediate nodes that are not defined must be set to 0. If only 4 nodes are to be given the user may just specify those 4 and the rest will be set to 0. To exit from the DIGELE command the user may digitize the same point three times or pick the 'R' or 'E' blocks in the menu area (if the digitize option is selected).

2.3.5 DISPLAY VALUES

The user may execute the DISPlay command to obtain information concerning the status of the model being defined and user definable parameters that affect model data definition. Figure 2.22 illustrates the output provided by the DISPlay command. All model parameter values are calculated in response to the command, so repeated use of the command can be wasteful. CORGEN will automatically provide this information before program termination.

2.3.6 ELEMENT DATA

The ELEMent command provides the user with capabilities for implementing the mesh generation features of CORGEN. Options exist for selecting alternate groups of nodes to be utilized in the element generator or for the user to enter new elements directly from the keyboard. Editing features are provided which allow the user to alter nodes specified in element connectivity or to replace or delete elements. Elements can be listed by the selection of a range. Figure 2.23 illustrates the command options available with the ELEMent command. The nine node biquadratic

```
MODEL SUMMARY DATA:

NUMBER OF NODES DEFINED.......:
HIGHEST NUMBERED NODE.........:
NUMBER OF UNDEFINED NODES.....:

NUMBER OF GROUPS DEFINED......:
HIGHEST GROUP DEFINED.........:
NUMBER OF UNDEFINED GROUPS....:

NUMBER OF ELEMENTS DEFINED....:
HIGHEST ELEMENT DEFINED.......:
NUMBER OF UNDEFINED ELEMENTS..:

NUMBER OF ELEMENT REFERENCES
TO UNDEFINED NODES............:
```

Figure 2.22. DISPlay Command Output listing

ELEMENT - specify element generation editing

ALTER - change element connectivity

GENERATE - generate elements using selected groups

FILE - input element connectivity from file

KEYBOARD - input element connectivity from keyboard

HELP - help user

INTERPOLATE - refine mesh by interpolation

LIST - list elements

DELETE - delete elements

TABLE - give summary of element statistics

Figure 2.23. ELEMent Command Options.

element illustrated in Figure 2.21 is the element type generated by the ELEMent command. The program allows the user to specify element connectivity directly, however, the user must specify elements that correspond to the local node numbering given in Figure 2.21 for the quadratic surface element.

If the INTErpolate option has not been invoked at the time elements are generated, CORGEN permits all existing nodal data to be interpolated (refined) in both surface directions) prior to generating elements. Invoking the interpolation option at this time produces twice as many elements in each direction (refinement may also be postponed until PREP is executed; see Subsection 4.1).

2.3.7 FINISH MODEL

Once the model nodal coordinate data has been input properly to form appropriate groups, the user may select the FINIsh command to complete all element generation and preprocessor functions to conclude the program. This command will access the current groups list whether established by default or selected by the user, and invoke any necessary interpolation procedures before executing the element generator. The command writes an unformatted data file which the user may input directly to the EXPAND module to obtain the 3-D solid element model needed for input to the PREP program. This command will provide a listing of the model parameters illustrated in Figure 2.22 once the model has been completed. Included in this listing will be the file number containing the current model data. The STOP command has the same effect as FINISh.

2.3.8 GROUP DATA

Groups are employed by CORGEN as an intermediate organizational structure to facilitate element generation. The GROUp command options, illustrated in Figure 2.24, allow the user to alter the node ranges defined in the groups, to list defined groups, to select alternative groups or order of groups for element generation and to define new groups for use in element generation or new node interpolation. The concept of the group consisting of cross-sections of some model, precludes the definition of any node in more than one group specified for element generation. The user may, however, create new groups with existing nodes for the purposes of interpolating new nodes. The new groups may contain greater or fewer nodes than the original groups and will replace the original group sent to do the interpolation.

CORGEN provides interactive prompts to insure that the user properly defines new groups or makes valid changes to existing groups. Groups are defined by a beginning node, an ending node and a group designator. The group designator assists the user in keeping track of how the groups were created. Table 2.2 lists the different group designators that are assigned and defines their meanings. The current options for CORGEN assume that the user is able to always input or generate nodes such that functionally related nodes are adjacent to one another. Special cases may arise where the user must specify random lists of nodes for interpolation to yield a new group with the proper characteristics for the model to be defined.

2.3.9 INTERPOLATE DATA

CORGEN provides the user with a means of creating new coordinates based on the model data provided. The INTErpolate command allows users to input either a group of nodes or a list of nodes to use for the creation of new nodes. The input for the interpolation procedures requests the user to specify what nodes or group to use and how many nodes the new group being created should have. CORGEN will interpolate the given nodes

GROUP - select group specification and editing

ALTER - change group composition

INTERPOLATE - generate nodes with selected group

KEYBOARD - keyboard input of group composition

HELP - help user

LIST - list groups

DELETE - delete groups

TABLE - give summary of group statistics

Figure 2.24. GROUP Command Options.

TABLE 2.2 - CORGEN GROUP DESIGNATORS

Groups will be labelled depending on the type of input used to define the nodes to the program. The following group designators are used:

A - arc generator
C - curve generator
D - deleted group
F - file input
I - interpolated
K - keyboard input
L - line generator
S - screen digitizer
T - tablet digitizer

and provide a new group of nodes if no group was specified or it will overwrite the group that was given as input. The two end point nodes of the group will be included in the new group to insure the model boundaries are not affected. While the new group will replace the old group, new node numbers must be generated for the nodes created by interpolation. These new node numbers in no way correspond to the old node numbers. The actual nodal coordinates used to do the interpolation are in no way affected by the generation of new nodes.

2.3.10 NODE DATA

CORGEN is structured to provide the user with several node input features which operate independently and concurrently. The user may access a FILE input of nodal data, then enter the KEYBoard option to add additional coordinate data. In addition, the digitizing command may be executed at any time to add lofting type data coordinates. The node input routines will prompt the user to declare how many nodes are to be included in the next group. If appropriate, the user also will be asked to specify how many groups are to be created before program control will be returned to the user. The input routine will then create or input enough nodes to equal the number of nodes per group times the number of groups. The groups created may then be utilized to create other groups of nodes or to specify how nodes should be organized for element generation. Figure 2.25 lists the options for the NODE command.

All nodal coordinate data is stored as four values. Upon input to CORGEN all data is converted to rectilinear coordinates and stored as (X,Y,Z) triples for each coordinate location. In addition, a thickness parameter is associated with each coordinate set for use in EXPANDing the surface model to a solid model prior to use by the PREP preprocessor. The user may specify a default thickness value which will be appended to all

NODE

ALTER - change nodal attributes

GENERATE - create new nodes using automatic generators

FILE - input nodes from a file

KEYBOARD - input nodes from keyboard

HELP - help user

LIST - list nodes

DELETE - delete nodes

TABLE - provide a table of nodal statistics

INTERPOLATE - create new nodes via interpolation

Figure 2.25. NODE Command Options.

the coordinate data input until the user resets it. The ALTEr option also permits the user considerable flexibility in making changes to attributes (e.g. - X,Y,Z,THICK) of a range, list or group of nodes. New nodes may be input to CORGEN via the ALTEr mode, however, all nodes entered in this fashion will be placed in a single new group. A new group will be created each time the user leaves the single node edit option and one or more previously undefined nodes were entered.

KEYBoard input of nodes allows the user to input nodes in coordinate triples plus a thickness value. Any coordinate transformation can be specified for the data as well as a separate coordinate translation. Lofting planes cannot be used with the keyboard input. All data for KEYBoard input is read format-free, meaning the user may type the data by separating each value with a comma or blank.

FILE input provides the user with a convenient means to input coordinate data already available on cards, tapes or disk files. The user may input data as coordinate triples plus a thickness value or may select to specify a default thickness and just input the coordinate triples. Again, the coordinate transformations may be selected for spherical or cylindrical coordinates if necessary. The user must be careful to specify how many groups are to be constructed by the current READ option. The command will request the user to specify the number of nodes that are to be included in each group being built then the number of groups to be constructed. The number of nodes on the file must be at least the number of nodes per group times the number of groups to build.

The DIGItize command allows the user to create digitizing planes arbitrarily located in three-dimensional space. The digitizing planes are defined either by providing the equation of the plane, three non-collinear points or three nodes currently defined that lie within the plane. Once the lofting plane has been defined the user may begin digitizing coordinates into groups, where each loft plane will generally correspond to a separate group. The command assumes a Cartesian coordinate system for all digitizing, although the user may select coordinate translations to perform on the data prior to the transformation from the digitizing plane to global coordinates. The user may exit the digitizing mode in two ways, one is to select the menu option and digitize the EXIT option in the menu area. The second manner is to digitize the same point three consecutive times. In the latter case the last three points will all be lost.

2.3.11 RESTORE DATA

Data that has been previously generated by CORGEN, AGRID or SPATCH in the unformatted, surface model file structure may be submitted to the CORGEN program with the RESTore command. This command will cause the current model in the CORGEN system to be deleted and the new model to replace it. Once the command has been issued the user will be prompted to input the appropriate data file specifier (see Appendix C for valid file names). CORGEN will then input the new nodes and elements and create groups for user convenience.

2.3.12 SAVE DATA

Data input to CORGEN or created by CORGEN may be saved at any time by the user with the SAVE command. This command will cause all currently defined nodal coordinate data, element connectivity data and groups specification data to be written to an unformatted surface model data file.

2.3.13 STOP PROGRAM

The STOP command will terminate CORGEN program execution. Program processing flags will be checked to insure the user has not overlooked any of the crucial operations necessary for model generation. If any phases of model development have not been accomplished the program will perform any operations that are necessary to produce a complete model file.

2.4 AGRID - Arbitrary Arrays of Points Mesh Input

The AGRID data entry program is designed for use with data that is unsuitable for use with the remaining data input modules, usually due to a lack of order within the geometric data itself. AGRID can accept a virtually arbitrary collection of point coordinates which lie on a surface of interest, and generate a regular grid of mesh points which define a smooth surface passing through the given data.

The method of surface-fitting used within AGRID is based upon a minimum-curvature criterion, by which the given data (in the form of point coordinates X, Y, Z) is used to define a continuous surface Z = Z(X,Y) which exhibits a minimal amount of oscillation between the known points. In particular, the final surface is represented by a grid of piecewise bicubic functions whose coefficients are determined by minimizing the curvature functional

$$\Phi = \iint_A \left[Z^2_{,xx} + Z^2_{,yy} + (Z_{,xx}Z_{,yy} - Z^2_{,xy}) \right] dx\, dy \qquad (2.2)$$

which reflects both the mean and Gaussian curvatures of the surface. The minimization is carried out by solving a small finite element problem, in which each bicubic segment of the surface corresponds to a single finite element.

Figures 2.26 through 2.28 illustrate the AGRID solution procedure. The known points lying on the surface to be defined are enclosed within a rectangular region which will form a temporary grid for interpolation (Figure 2.26). A mesh of 'interpolation elements' is defined by specifying the series of X (or Y) coordinates at which mesh points are to be positioned, as shown in Figure 2.27. The mesh size and refinement are arbitrary, but more divisions would normally be used in those regions containing

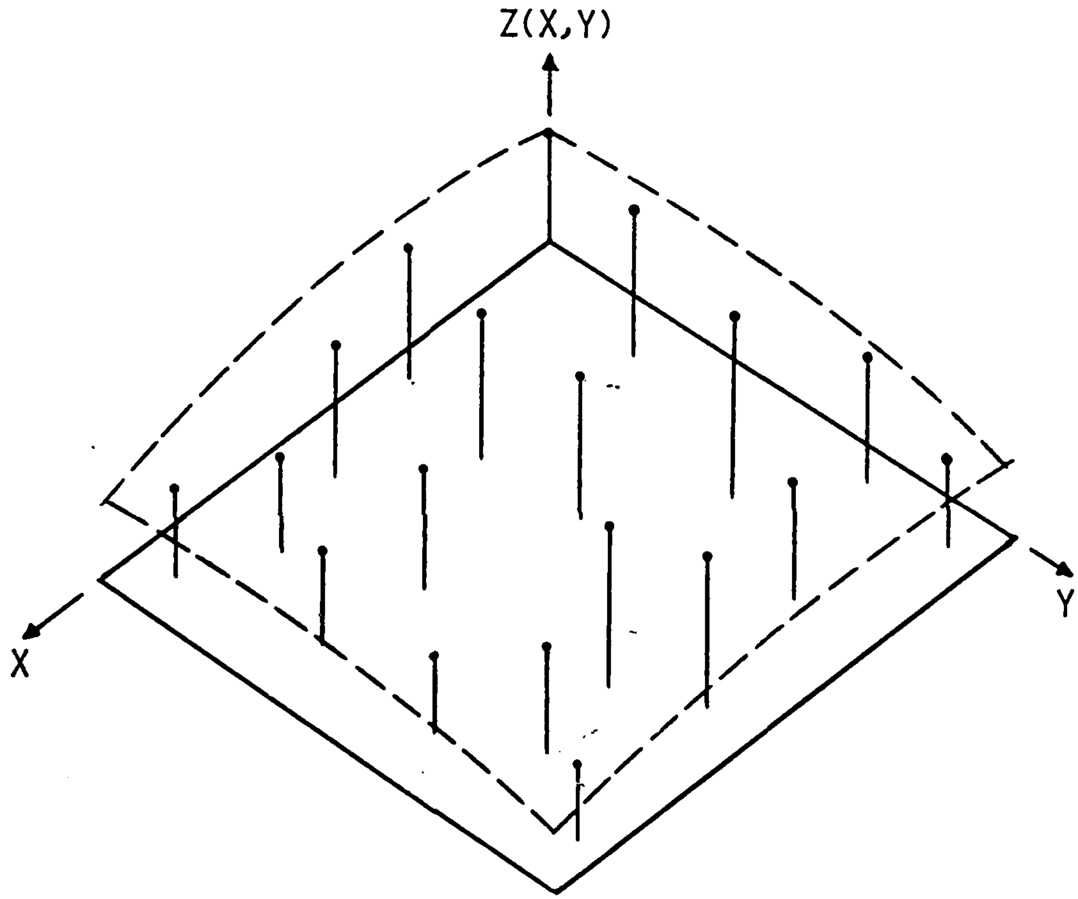


Figure 2.26. Arbitrary Array of Surface Points.

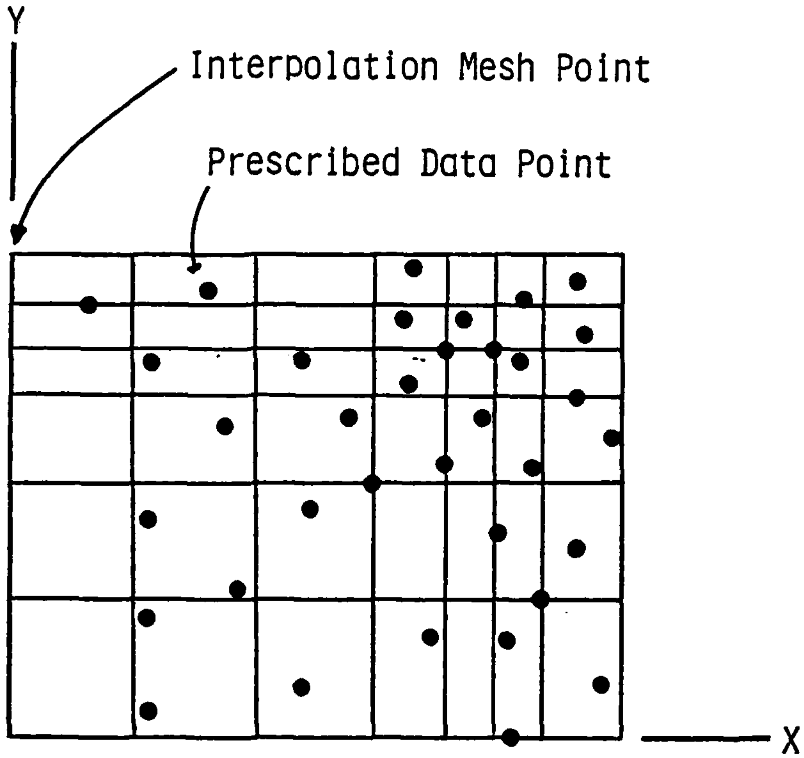


Figure 2.27. AGRID Interpolation Grid.

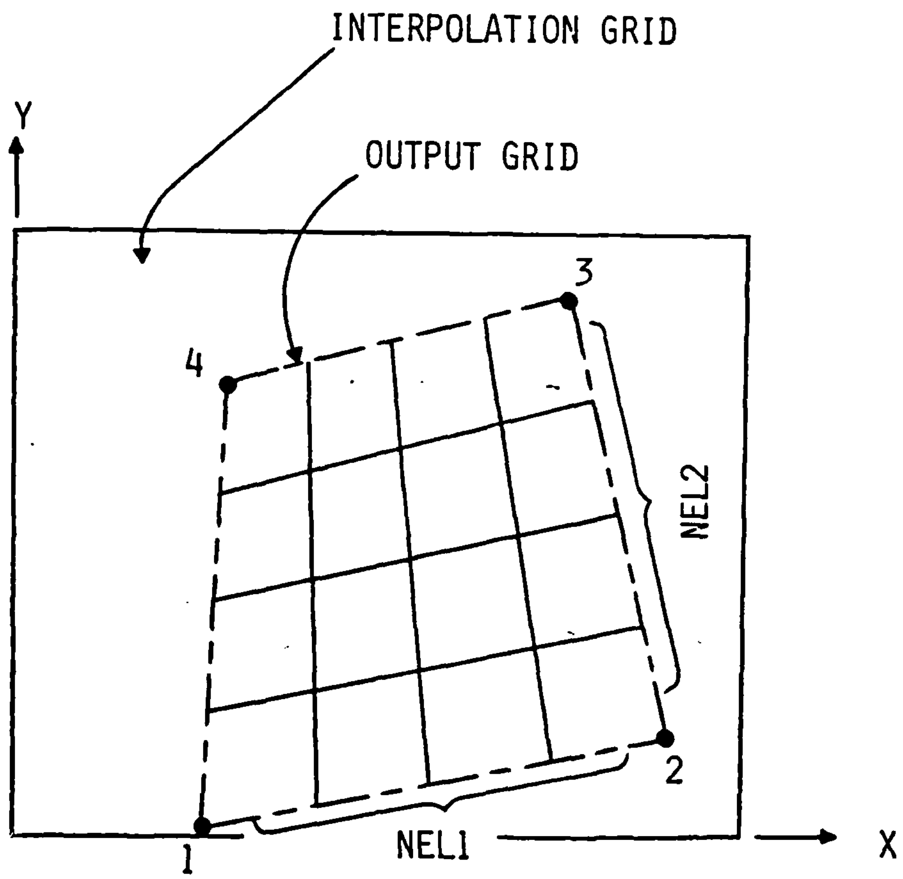


Figure 2.28. Boundary Definition for Output Grid.

the largest number of prescribed coordinates or exhibiting the most changes in curvature. The coordinates of known points on the surface are then specified; these may be located anywhere within the interpolation grid, but not outside it. Finally, a set of boundaries is defined which outline the true limits of the surface mesh to be generated, and the number of surface elements to be generated along each direction of the final mesh is specified. The projection of these boundary curves on the (X,Y) plane is a straight-sided quadrilateral, as shown in Figure 2.28. The result of this procedure is a mesh of surface elements (see, for example, Figure 2.2) and coordinates, suitable for input to EXPAND.

The necessary input to AGRID is summarized in Table 2.3. This data is normally saved on a file (local file INGEOM, on CDC machines) arranged in free format, with spaces or commas delimiting separate items of data within a record. The output from AGRID, in addition to a minor amount of printed output, is in the form of the shell surface geometry file described in Subsection 6.1 and in Appendix A.

It should be observed that AGRID is based upon an empirical means of interpolation, due to the generality of the data which can be accepted. In return for this generality, a compromise in the quality of the interpolation can result if the amount of data given is insufficient to define the surface with a fair degree of accuracy. For example, the specification of three data points to AGRID will always result in the generation of a planar surface, even if the intended result is a curvilinear surface; four points will generate a warped surface with small mean curvatures. When the local changes in curvature are significant, sufficient data must be specified to make them evident to the generator. If this advice is followed, AGRID is capable of producing very nice results from the most disorganized data.

TABLE 2.3 - AGRID Data Input

RECORD	VARIABLE	DESCRIPTION
1	NXINT	Number of X-stations in rectangular interpolation grid
	NYINT	Number of Y-stations in rectangular interpolation grid
	NSPEC	Number of prescribed coordinate values to be given
2	X(NXINT)	Array of X-station values, in ascending order
3	Y(NYINT)	Array of Y-station values, in ascending order
4	XVAL	X-Coordinate for prescribed point
	YVAL	Y-Coordinate for prescribed point
	ITYPE	Type of value prescribed (=1 for Z, =2 for dZ/dX, =3 for dZ/dY)
	VALUE	Value of Z, dZ/dX, or dZ/dY at point (XVAL,YVAL)
<<< Repeat record 4 for each prescribed point (NSPEC times) >>>		
5	XOUT(4)	X-Coordinates of corner points of output grid
6	YOUT(4)	Y-Coordinates of corner points of output grid
7	NEL1	Number of elements in output grid, along edge 1-2
	NEL2	Number of elements in output grid, along edge 2-3
8	THICK	Uniform thickness value for use in EXPAND

2.5 SPATCH - Surface Patch Geometry Input

Recently, many interactive surface design systems have adopted the bicubic patch as a basis for geometric description. The SPATCH utility is a special-purpose conversion program which, with the aid of a few simple user-written subroutines, allows the translation of bicubic surface patch data into the preprocessor data format.

With the type of surface patch considered in SPATCH, all three Cartesian coordinates X, Y, Z are described as parametric functions of two natural coordinates (u,v),

$$
\begin{aligned}
X &= X\,(u,v) \\
Y &= Y\,(u,v) \qquad (2.2) \\
Z &= Z\,(u,v)
\end{aligned}
$$

where u and v each vary between zero and one. The most common form of storing the surface patch data is in arrays having the form [7]

$$
\left[\begin{array}{cc|cc}
X_{00} & X_{01} & X_{v00} & X_{v01} \\
X_{10} & X_{11} & X_{v10} & X_{v11} \\
\hline
X_{u00} & X_{u01} & X_{uv00} & X_{uv01} \\
X_{u10} & X_{u11} & X_{uv10} & X_{uv11}
\end{array}\right] \qquad (2.3)
$$

for each of the coordinates X, Y, Z. The indices (o,l) refer to the values of u and v respectively, and subscripts u, v denote parametric derivatives of the coordinate in question. For example, Xvlo is the value of dX/dv at u=1 and v=o.

SPATCH permits surfaces described in the form of a collection of such patches to be translated into an assemblage of biquadratic surface elements (see Subsection 2.1, Figure 2.2) suitable for input to EXPAND. The input to SPATCH is performed through a user-written subroutine, since the actual storage of the surface patch data may taken on a number of different forms. The user routine has the form

```
    SUBROUTINE UPATCH (PATCH, THICK, NU, NV, ICOUNT, IEND)
    DIMENSION PATCH(4,4,3)
       .        .
       .        .
    < code to read surface patch data >
       .        .
       .        .
    RETURN
    END
```

The exact meanings of all formal parameters and other special requirements are discussed in Appendix E. Each surface patch (described by the coefficients in array PATCH) may be divided into one or more nine-node surface elements, as determined by the parameters NU and NV.

Two additional user-written subroutines are provided for in SPATCH, to permit opening and/or positioning of the patch data file prior to entering UPATCH, and closing of the file upon completion. These two routines will generally be necessary only in the VAX version of SPATCH, or in the event that other, extraneous data precedes the surface patch data on the file to be accessed. These two file control subroutines, UOPEN and UCLOSE, are also described in Appendix E.

SECTION 3

PREPROCESSOR INTERFACE MODULES

The translation of existing finite element model data into the standard preprocessor format is accomplished using special-purpose data translation utilities, each one corresponding to a specific pair of data formats. Conversion of modelling data into the standard PREP format is currently possible for two data file types:*

- MAGNA [4] finite element data deck; and
- IMPRESS [3] preprocessor database.

These two data conversion utilities are described in subsections 3.1 and 3.2, respectively.

In addition to the special-purpose translation programs mentioned above, a set of general-purpose utility subroutines exist for the purpose of converting finite element data in other external formats. These general interface routines are the subject of subsection 3.3.

---

*Conversions to and from the PREP file format can also be performed using the NEUTRAL interface program (Section 5). NEUTRAL is used primarily for archival of data in formatted form.

3.1 TRNSFR - MAGNA INPUT DATA TRANSLATOR

The TRNSFR data translator accepts as input the standard data deck recognized by the MAGNA finite element program [4]. Only the nodal coordinates and element connectivity data, which represents the bulk of the model information, are converted to the preprocessor internal format. All of the standard MAGNA element types are supported by TRNSFR; these include:

- truss (bar) elements, with two nodes per element,
- beam elements, with two or three nodes per element,
- plane stress, plane strain and shear panel elements having from four to nine nodes per element,
- axisymmetric elements with four to nine nodes,
- the MAGNA thin shell elements, with eight or sixteen vertex nodes, and
- three-dimensional solids, with from eight to 27 nodes.

TRNSFR is capable of accepting all of the incremental node point and element generating sequences possible with the MAGNA program.

Execution of TRNSFR is quite simple (see Appendix C). On CDC computers, the MAGNA input deck must be attached as a local file prior to execution, while on the VAX 11/780, the data file name is prompted by the program. No further input or action is required of the user, other than saving the generated preprocessor data file.

3.2 IMPRINT - IMPRESS PREPROCESSOR INTERFACE

IMPRESS is the main preprocessing program of a finite element data generation system consisting of three distinct programs (IMPRESS, INTRPOS, and IMPACT), all of which are described in Reference [3]. The modelling features available in IMPRESS include a flexible area-outlining approach to geometry definition, and the automatic calculation of intersection curves for planar, cylindrical and spherical shapes. Both two- and three-dimensional geometric models may be prepared with the program.

Finite element data originally generated using IMPRESS can be converted for use with the present preprocessor by means of the IMPRINT translator. IMPRINT is a collection of conversion routines which can be used as the output overlay of the IMPRESS data base access program INTRPOS. The IMPRINT overlay is merged with INTRPOS using the control statement sequence described in Appendix I of Reference [3]. When INTRPOS is executed, the data conversion procedure is invoked by entering

```
PREPARE RAW
```

Upon completion of the data conversion, the END directive can be used to exit INTRPOS. The converted finite element data is written to the local data file TAPE9, which should be saved following execution of the program.

Table 3.1 summarizes the IMPRESS data types, and indicates those which are supported by the IMPRINT utility.

TABLE 3.1 - IMPRESS Data Types

IMPRESS Data Type	Supported by IMPRINT
- Nodal Coordinates	YES
- Skewed Coordinate Systems	NO
- Boundary Conditions	YES
- Master / Slave Constraints	NO
- Prescribed Displacements	NO
- Element Connectivity	YES
- Element Property Codes	YES
- Beam Section Properties	NO
- Concentrated Springs / Masses	NO
- Nodal Concentrated Forces	YES
- Multiple Loading Sets	YES

3.3 GENERAL-PURPOSE INTERFACE ROUTINES

A collection of general-purpose translation routines has been developed to facilitate the conversion of foreign data formats to the standard internal format of the present preprocessor. To use these interface routines, the user must write a small driver program which reads the data to be converted, and sends it to the translation subroutines for consistency checking and output. Seven utility routines are available:

1. INICNV - Initialization routine
2. NODCNV - Nodal data conversion
3. ELECNV - Element data conversion
4. BCDCNV - Boundary condition data conversion
5. NLDCNV - Nodal loads data conversion
6. ELDCNV - Element distributed loads data conversion
7. TRMCNV - Termination routine.

These interface routines are described in detail in Tables 3.2 through 3.8.

It is important to note that logical unit 5Ø must be reserved for output by the interface routines, and should not be modified by the calling program. On CDC machines, the file TAPE5Ø must be declared on the PROGRAM card, with a minimum buffer length of 512 words. For the VAX version of the conversion routines, the output

TABLE 3.2

INICNV - Initialization Subroutine

PURPOSE - Define the number of data entries to be made in the file, and initialize internal parameter values.

ACCESS - CALL INICNV (NUMNP,NUMEL,NUMBC,NUMNL,NUMEF)

PARAMETERS -

NUMNP	(input)	- Number of nodal points in the model
NUMEL	(input)	- Number of finite elements
NUMBC	(input)	- Number of boundary condition specifications
NUMNL	(input)	- Number of nodal loads specifications
NUMEF	(input)	- Number of element loads specifications

NOTES - (1) This subroutine must be called before any of the other data translation routines are called.
(2) Nodal points must be numbered from 1 through NUMNP.
(3) Each boundary condition specification may consist of a range of nodes (first, last, increment) all having the same constraints applied.
(4) Each nodal load specification may consist of a range of nodes (first, last, increment) and three components of force (X, Y, Z) which apply to all nodes in the sequence.
(5) Each element load specification may consist of a range of elements (first, last, increment) and a loading type and value which apply to all elements in the range.

TABLE 3.3

NODCNV - Nodal Coordinate Conversion Subroutine

PURPOSE - Process coordinate data for a single node point of the model.

ACCESS - CALL NODCNV (NODE,X,Y,Z)

PARAMETERS -

NODE	(input)	- Node point sequence number
X	(input)	- Cartesian X-Coordinate
Y	(input)	- Cartesian Y-Coordinate
Z	(input)	- Cartesian Z-Coordinate

NOTES - (1) Node points must be transmitted to NODCNV sequentially, with all node numbers between 1 and NUMNP.
(2) Node coordinates must be defined prior to defining elements, boundary conditions or loading data.

TABLE 3.4

ELECNV - Element Data Conversion Subroutine

PURPOSE - Processes a single finite element of the model.

ACCESS - CALL ELECNV (IELNUM,MATLNO,IDIMEN,MAXNOD,NCON)

PARAMETERS -

IELNUM	(input)	- Element number (arbitrary)
MATLNO	(input)	- Material number (optional)
IDIMEN	(input)	- Dimensionality of the element (1,2,3)
MAXNOD	(input)	- Maximum local node number
NCON	(input)	- List of connected nodes

NOTES - (1) If the material number is to be ignored, set MATLNO=Ø.
(2) MAXNOD should lie within the following ranges:

MAXNOD = 2	- One-dimensional elements;
MAXNOD = 4 - 9	- Two-dimensional elements; and
MAXNOD = 8 - 27	- Three-dimensional elements.

(3) The length of the connection array NCON is determined by MAXNOD. If variable-number-of-nodes elements are used, intermediate entries for which no node exists must be entered as zeroes.
(4) The conventions for node numbering in all element types are shown in Figures 2.1 and 2.2.

TABLE 3.5

BCDCNV - Boundary Condition Conversion Subroutine

PURPOSE - Processes a single boundary condition specification.

ACCESS - CALL BCDCNV (ISTART,IEND,INCR,IX,IY,IZ)

PARAMETERS -

ISTART	(input)	- First node in the sequence
IEND	(input)	- Last node in the sequence
INCR	(input)	- Node number increment
IX	(input)	- X-Direction constraint code (Ø,1)
IY	(input)	- Y-Direction constraint code (Ø,1)
IZ	(input)	- Z-Direction constraint code (Ø,1)

NOTES - (1) The specified constraints will be applied at nodes ISTART, (ISTART+INCR), (ISTART+2*INCR), ... , IEND. If a single node is to be constrained, IEND and INCR may be set to zero; however, they must appear in the calling statement.

(2) IX, IY, and IZ are defined as zero if motion is permitted in the X, Y, or Z directions, respectively. Constraints are specified by IX, IY, or IZ = 1 as appropriate.

TABLE 3.6

NLDCNV - Nodal Loads Conversion Subroutine

PURPOSE - Processes a single nodal loading specification.

ACCESS - CALL NLDCNV (ISTART,IEND,INCR,FX,FY,FZ,ICASE)

PARAMETERS -

ISTART	(input)	- First node number in sequence
IEND	(input)	- Last node number in sequence
INCR	(input)	- Node number increment
FX	(input)	- X-Direction force component
FY	(input)	- Y-Direction force component
FZ	(input)	- Z-Direction force component
ICASE	(input)	- Load case or group number

NOTES - (1) The range of nodes to receive the specified load is defined exectly as in the boundary condition data; when only a single node is to be loaded, IEND and INCR may be set to zero.
(2) Any integer value may be used for the load case number.

TABLE 3.7

ELDCNV - Element Loads Conversion Subroutine

PURPOSE - Processes a single distributed element load specification.

ACCESS - CALL ELDCNV (ISTART,IEND,INCR,ITYPE,ICASE,FORCE)

PARAMETERS -

ISTART	(input)	- First element number in sequence
IEND	(input)	- Last element number in sequence
INCR	(input)	- Element number increment
ITYPE	(input)	- Loading type/ direction/ surface code
ICASE	(input)	- Load case or group number
FORCE	(input)	- Loading magnitude

NOTES - (1) Element ranges are defined similarly to the nodal ranges for boundary conditions and nodal loads. If a single element is to be loaded, IEND and INCR may be set to zero, but they must appear in the calling statement.

(2) ITYPE is defined as follows:

ITYPE = -1, -2, -3 signifies a body force (force per unit volume) in the X, Y, or Z direction, respectively.

ITYPE = 1, 2, ... , 6 denotes a surface pressure on element faces 1, 2, ..., 6, respectively, for three-dimensional elements (IDIMEN=3).

ITYPE = 1, 2, 3, 4 represents a line load (force per unit length) on the corresponding edge of a two-dimensional element (IDIMEN=2).

TABLE 3.8

TRMCNV - Termination Subroutine

PURPOSE - Checks for incomplete blocks of data, and closes the generated data file.

ACCESS - CALL TRMCNV

PARAMETERS - (none)

NOTES - (1) In the CDC version of the termination routine, the output data file is rewound, but not returned.

file generation is user-transparent, with the converted data being written automatically to the file UNFMT.DAT. A labelled COMMON block (COMMON /DATCNV/) is also used in the interface subroutines, and should not be modified by the user.

SECTION 4

MODEL VERIFICATION AND MODIFICATION

Capabilities for developing a completed finite element model from an initial geometric description are contained in the central program, PREP. Typical operations performed within PREP are mesh refinement, assignment of properties, constraints and loads, and geometry plotting. In the following subsections, each modelling operation available in the PREP preprocessor is described, and typical sequences of operations are described which make the most effective use of the program's capabilities.

The general mode of operation of the PREP module involves the use of model data files, with up to 13 files being active at any one time. Each file may be named for later reference. The typical PREP operation then consists of the sequence

```
INPUT MODEL ---> (OPERATION) ---> OUTPUT MODEL
  FILE(S)                           FILE(S)
```

For example, the user may wish to refine the finite element mesh in a certain area of a model. Upon entering the command 'REFINE', he is requested to enter the name of an existing model whose element mesh is to be refined. PREP then requests additional input to define those elements to be subdivided and the degree of refinement, and generates a new model file (which is assigned a new name) with the requested element refinement.

The input/output files for the PREP preprocessor are described in Section 6.1 and in Appendix A. PREP is organized such that data files may be manipulated in the program, saved on peripheral storage, and reentered into PREP at a later time. This capability permits the generation of a model to be accomplished in several sessions if necessary, and allows separate segments of the finite element model to be prepared separately and combined when complete.

The various modelling functions available in PREP are described in Subsections 4.1 through 4.7; those utilities which are useful in managing data files within the program are also discussed in Subsection 4.1. Subsection 4.8 outlines some suggested sequences of PREP operations which have proved to be most effective in practice. Subsection 4.8 also contains an alphabetized listing of all PREP commands for quick reference.

4.1 MODIFICATION / REFINEMENT OF MODEL GEOMETRY

PREP operations which are provided for manipulation of the geometry of a finite element model are:

- CREATE - build model from 'scratch' by user input;
- EDIT - alter coordinates and connectivity;
- FILL - generate midside and interior node points;
- MASK - remove midside and interior node points;
- MERGE - combine two models with compatible geometries;
- REFINE - subdivide finite elements to form a finer mesh;
- REFLECT - form the mirror image of a model about a plane;
- ROTATE - rotate an entire model in three dimensions; and
- TRANSLATE - translate a model in three dimensions.

The CREATE and EDIT commands permit input, modification or deletion of individual items of modeling data. The remaining commands are geometric operations, which alter either the coordinate data or the element connection data for the entire model in question. The effect of each of these commands, as well as the available options for data input, are summarized in Tables 4.1 through 4.7. An example of each operation is given in the accompanying Figures (4.1 through 4.4).

It is generally advisable to perform FILL and/or MASK operations at the beginning of a modelling session, particularly in cases where an irregular number of nodes has been used in the initial generation of a model. An example in which REFINE has been used prior to FILL with such an irregular mesh, is shown in Figure 4.5; here the initial geometry has been defined using elements with only one midside node, and the effect of the REFINE operation is disastrous. The remaining functions (MERGE, REFINE, REFLECT, ROTATE and TRANSLATE) can normally be invoked in any order as circumstances dictate.

TABLE 4.1 - The FILL Command

Command	: FILL
Effect	: Generates midside and/or interior nodes to create higher-order finite elements.
Number of Input Files	: 1
Number of Output Files	: 1
Options	: (1) All 3-D elements FILLed to 16, 2Ø, 26 or 27 nodes (2) All 2-D elements FILLed to 8 or 9 nodes
Related commands / options	: MASK is an 'inverse' operation to FILL.
Notes	: This function does not preserve boundary condition and loading data.

TABLE 4.2 - The MASK Command

Command : MASK

Effect : Removes midside and / or interior nodes from elements.

Number of Input Files : 1

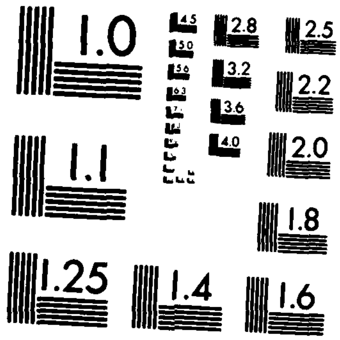
Number of Output Files : 1

Options : (1) All 3-D elements MASKed to 8, 16, 20, 26 or 27 nodes
(2) All 2-D elements MASKed to 4, 8 or 9 nodes

Related commands / options : FILL is an 'inverse' function to MASK.

Notes : (1) FILL does not preserve constraint or loading data.
(2) Care should be taken in using MASK with curved elements, since straight-sided elements will result in many cases.

AD-A129 025 MAGNA (MATERIALLY AND GEOMETRICALLY NONLINEAR ANALYSIS) 2/3
PART II PREPROCES..(U) DAYTON UNIV OH RESEARCH INST
T S BRUNER ET AL. DEC 82 UDR-TR-82-112
UNCLASSIFIED AFWAL-TR-82-3098-PT-2 F33615-80-C-3403 F/G 9/2 NL



MICROCOPY RESOLUTION TEST CHART
NATIONAL BUREAU OF STANDARDS-1963-A

TABLE 4.3 - The MERGE Command

Command	: MERGe
Effect	: Combines two models to form a single (output) model, with coincident nodes equivalenced and resequenced.
Number of Input Files	: 2
Number of Output Files	: 1
Options	: (None)
Related commands / options	: The TOLErance command can be used to modify the distance tolerance used in detecting coincident nodes.
Notes	: MERGE does not preserve constraint or loading data.

TABLE 4.4 - The REFINE Command

Command : REFIne

Effect : Subdivides selected elements of a model to produce a finer mesh for analysis.

Number of Input Files : 1

Number of Output Files : 1

Options : (1) Up to 150 elements may be refined at one time. The selection options include
- random element numbers
- ranges of elements (first, last, increment).

(2) Elements must be refined in one direction at a time. The refinement direction follows the natural coordinate directions of an element (see Figure 4.3). The PLOT option ORIENT should be used to determine the proper direction for element refinement.

(3) From one to five 'cuts' may be made through each element specified, resulting in from two to six elements as output. The default option is equal spacing of these cuts; by overriding the default, cuts may be spaced at irregular intervals across the elements.

Related commands / options : The PLOT utility can be used to identify the correct element numbers for input to REFINE (select the LABEL option), as well as the correct local directions (use the ORIENT option).

Notes : REFINE does not preserve boundary condition or loading data.

TABLE 4.5 - The REFLECT Command

Command : REFLect

Effect : Forms the mirror image of a model with respect to a specified plane in three-dimensional space.

Number of Input Files : 1

Number of Output Files : 1

Options : The plane of reflection may be defined in two ways:

- Give the coefficients of Ax + By + Cz = D; for example, the plane x = 5 has A = 1, B = Ø, C = Ø, D = 5.
- Define three non-collinear points which lie in the plane.

Related commands / options : REFLECT is often used in conjunction with MERGE to create a full model from a symmetric one.

Notes : (None)

TABLE 4.6 - The ROTATE Command

Command	: ROTAte
Effect	: Performs a sequence of three rotations of a model in space.
Number of Input Files	: 1
Number of Output Files	: 1
Options	: (None)
Related commands / options	: MERGE may be used following ROTATE to generate a full model from one for which a single sector (e.g., in polar coordinates) has been defined.
Notes	: Model rotations are always specified in degrees, and are always performed in the order: X-rotation, then Y-rotation, then Z-rotation.

TABLE 4.7 - The TRANSLATE Command

Command : TRANslate

Effect : Performs a rigid-body translation of a model in space.

Number of Input Files : 1

Number of Output Files : 1

Options : (None)

Related commands / options : MERGE is useful in conjunction with the TRANSLATE function when the geometry to be generated is periodic.

Notes : (None)

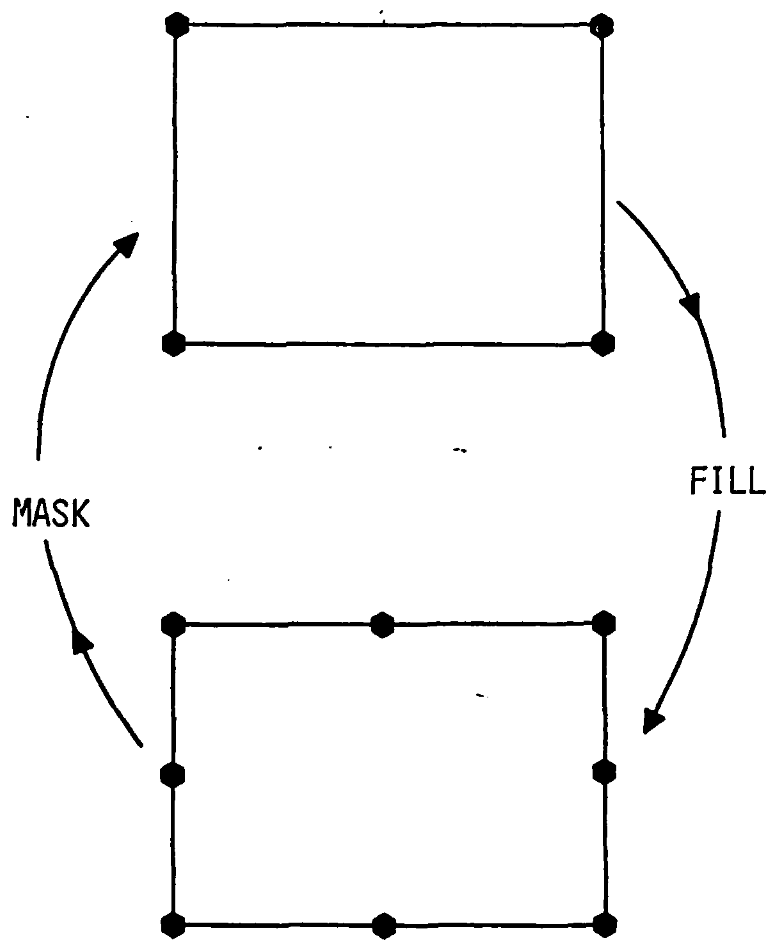


Figure 4.1. FILL and MASK Operations.

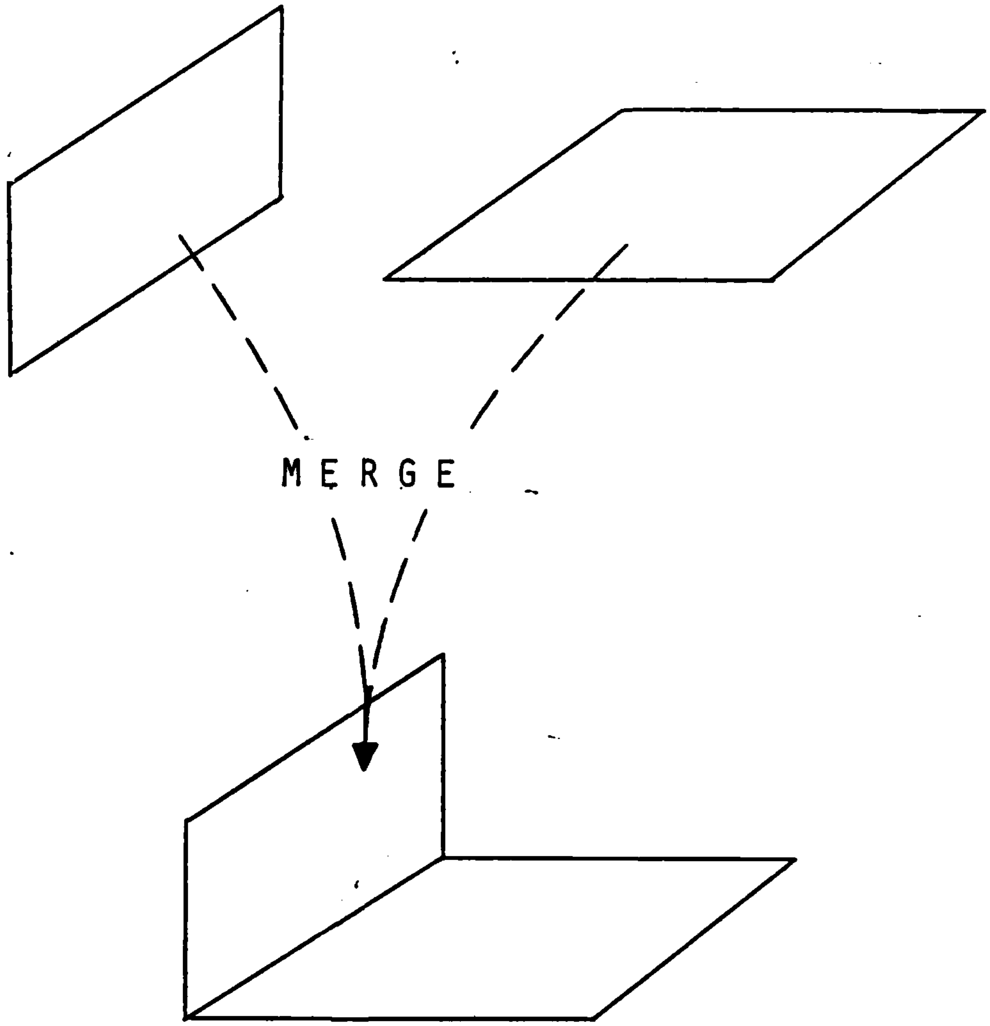


Figure 4.2. Effect of the MERGE Command.

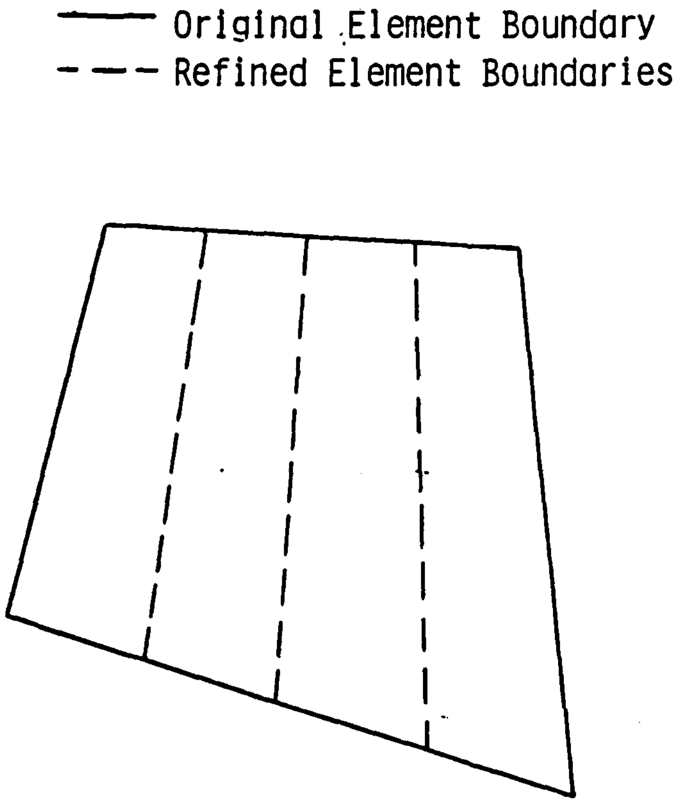


Figure 4.3. Effect of the REFINE Command.

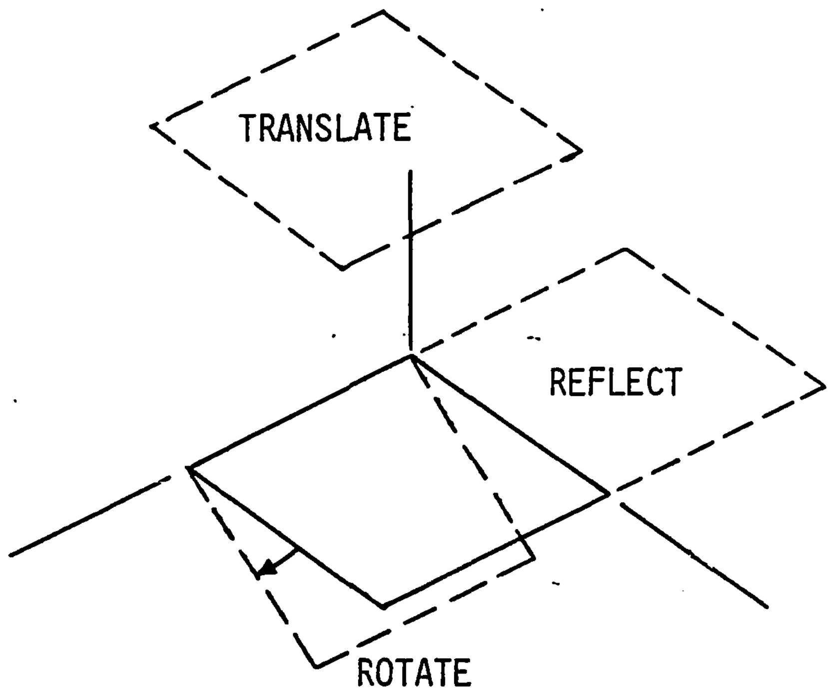


Figure 4.4. REFLECT, ROTATE and TRANSLATE Options.

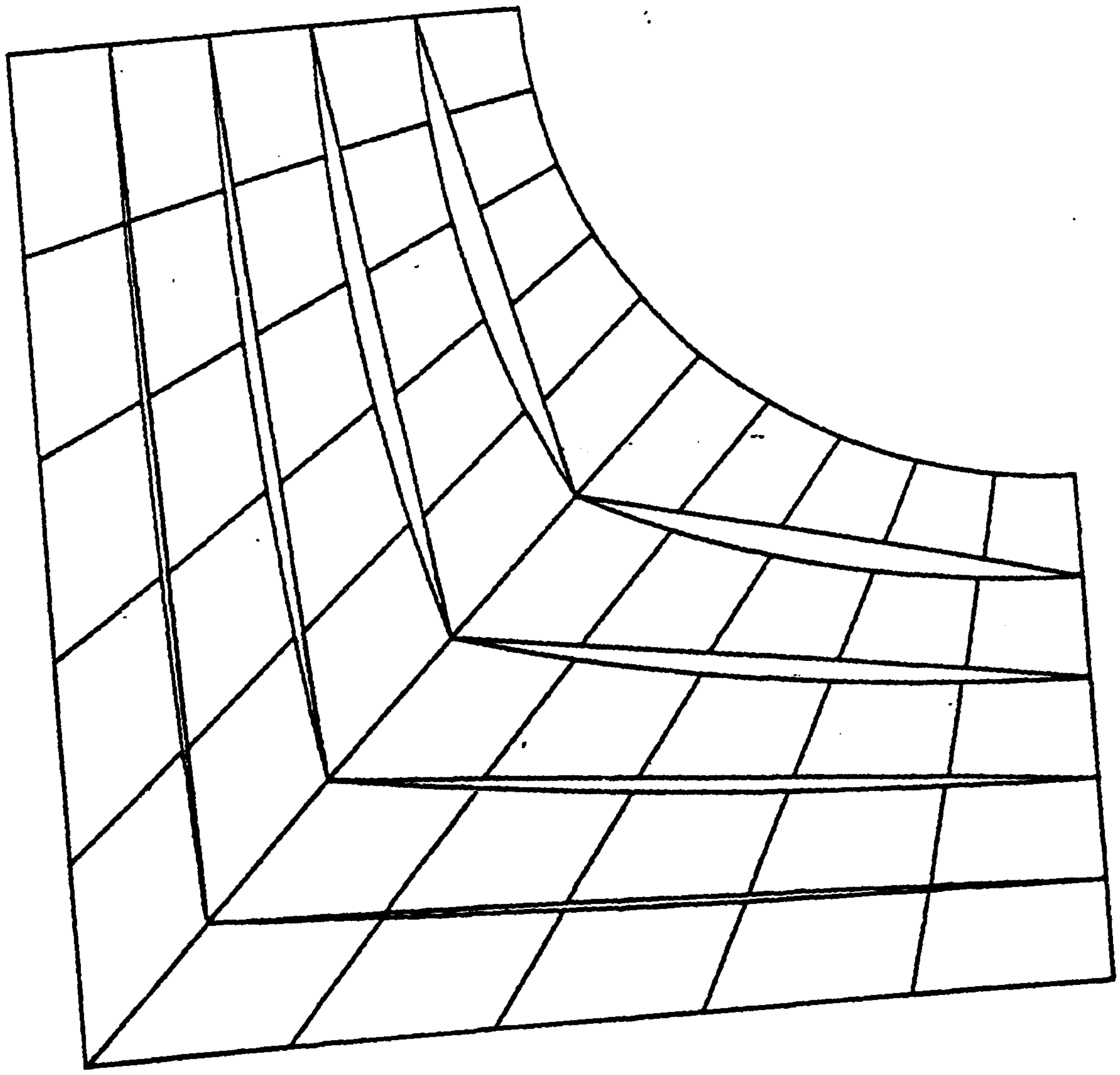


Figure 4 5. Eff<?t of the REFINE Operation Prior to FILL.

In addition to the above geometric modelling functions, two important auxiliary operations are included in PREP to ensure the correct generation of finite element data. They are

- SIFT - remove unused nodes from the model, and
- TIDY - combine all coincident nodes into single node points with a unique node number.

For models generated using the present preprocessor, these two utilities should be transparent: SIFT is called automatically following any MASK operation, and TIDY is performed whenever the FILL, MERGE, or REFINE commands are executed. However, geometric data from other sources may contain unused and / or duplicate nodes, in which case SIFT and TIDY may be executed as separate operations. Two options are available in the TIDY operation: OPT1 will generate correct results for any situation, but always requires more execution time; OPT2 uses a geometric partitioning of the model to reduce the complexity (and execution time) of the operation, but may fail in some instances due to storage limitations. If the faster TIDY option fails, the OPT1 version is invoked and the user is informed of the situation.

It should be noted that any geometric model may be listed and / or plotted at any time during the session using the PRINT and PLOT commands (Subsections 4.6 and 4.7). The PLOT option particularly useful before a REFINE operation, to identify the local directions (R,S,T) within an element which are needed as data for REFINE (see the ORIENT command, Subsection 4.6).

Additional auxiliary operations which are useful in managing model data files during REFINEment, MERGing and other operations include

- COPY - creates a copy of a model file, under a different name;
- DELETE - deletes the name of a model from the active list;

- HELP - gives a summary of all available commands, and further information on selected commands if requested;
- NAME - assigns a user-defined name to a model file;
- LIST - lists the names of all active model files;
- TIME - prints the CPU time elapsed since sign-on; and
- TOLERANCE - allows redefinition of the tolerances used in detecting coincident nodes in FILL, MERGE and REFINE.

The HELP and LIST commands are particularly useful as reminders of operations which are available and of model files already created during the session.

Features are provided in PREP to allow the creation of models from 'scratch'. The CREATE command provides a means for input of nodal coordinates and element connectivity to define a model. The model must be composed of 3-D solid elements containing at least 8 nodes per element. The EDIT feature allows the user to alter nodal coordinates or connectivity for any model in PREP. These commands are fully interactive and require all data to be entered in 3-D coordinates.

4.2 ELEMENT PROPERTIES SELECTION

Three facilities exist in PREP for the specification of finite element properties:

- PROPerty - define element types, material codes and numerical integration orders;
- SHELl - identify selected elements as thin shells; and
- CONTact - create surface contact elements from certain surfaces of existing three-dimensional elements.

The PROPERTY directive is the usual means of defining the pertinent property information for existing elements of the model. Elements may be assigned properties in groups (all 2-D or all 3-D elements, random lists of elements, or element ranges), with a single set of property specifications applying to all elements in the group. Elements may be defined as being:

Type 0 : Default (solid, plane stress, beam, or bar),
Type 1 : Thin shells,
Type 2 : Plane stress (membrane) elements,
Type 3 : Plane strain elements,
Type 4 : Axisymmetric solids,
Type 5 : Shear panels, or
Type 6 : Surface contact elements.

If no type is declared for an element, or if the number of nodes is inappropriate for the type assigned, the default element type for the existing nodal pattern will be assigned in the REFMT module (see Subsection 5.1). Integration orders are treated similarly, and generally the default values assigned in REFMT are acceptable.

Material property codes may be assigned within the PROPerty utility, using material codes appearing in the material properties library described in Appendix F. Materials data can also be assigned within the REFMT processor, and existing material coefficients may be edited as well (see Subsection 5.1).

The SHELL command performs a similar function to PROPERTY. Selected elements may be redefined as being thin shells or three dimensional solids, without specifying integration orders and material property codes.

The function of CONTact is complementary to that of PROPERTY: while the PROPERTY command allows existing elements to be defined as special elements to be used in surface contact analysis (see Reference [4]), CONTACT provides a means of creating new elements which are immediately declared to be surface contact elements. The new contact elements are created from specified faces of existing three-dimensional solids.

One other function of the CONTACT command is to create elements which lie on the faces of three-dimensional solids, whose properties are subsequently redefined using the PROPERTY command. This device can be useful when thin coatings (e.g., protective or damping layers) are to be applied over the surface of shell or solid elements, with the coating layer(s) to be treated as membrane (plane stress) finite elements.

4.3 BOUNDARY CONDITIONS AND CONSTRAINTS

Nodal constraints may be applied to a finite element model in PREP using the BOUNDS and LINEAR commands. Although boundary conditions may be defined at any time during the construction of a model, it is advisable to complete most of the geometric modeling operations, including node reordering, before the constraints are specified (see Subsection 4.8 for typical sequences of operations).

With the BOUNDS operation, homogeneous nodal constraints (i.e., displacement = 0) may be specified for groups of nodes selected by one of the following methods:

- select all nodes;
- select all nodes on a plane defined by Ax + By + Cz = D; or
- select all nodes in a given range (first, last, increment).

Each constraint definition begins with the selection of one of the above options (or selection of "no more boundary conditions"). PREP then permits all of the selected points to be fixed in any combination of the X, Y and Z directions. This cycle (select nodes, define constraint direction) continues until the selection option "no more boundary conditions" is specified. BOUNDS may be used as many times as required to define all simple boundary conditions for the model.

The LINEAR function allows the specification of simple forms of linear constraints at individual nodes. This form of constraint is required when the displacement at a node is constrained in a direction which does not coincide with the global X, Y, Z axis directions. Linear constraints are defined for one node at a time; first a node number is selected, and then the direction of constraint is defined. The direction of constraint is always defined by giving the plane in which motion is permitted at the node (i.e., the constraint is applied in the direction normal to this plane). The plane in which motion occurs can be defined either by giving the equation of the plane (in the form $Ax + By + Cz = D$), or by listing the coordinates of three points lying in the plane. As with BOUNDS, the LINEAR operation may be performed any number of times to define the required constraints.

4.4 APPLIED LOADING

Applied loadings of two types may be defined in PREP:

- concentrated nodal forces; and
- distributed element loads, including body forces, surface pressures and edge loads.

Applied loads input is initiated using the LOADS command, which may be given as many times as necessary in the construction of a model. Five basic options are available in the LOADS function.

They are:

- (N)odal loads input;
- (E)lement loads input;
- (L)ist currently defined loads data;
- (H)elp (print a list of available options); and
- (S)top loading data input.

These options may be executed in any order within the LOADS function.

When the nodal loads option (N) is selected, PREP will accept data defining concentrated forces acting at existing node points of the model. The specification of nodal loads includes the set of nodes at which the load is to act, a loading case number, and the three components of force (X, Y, Z) which act at the nodes selected. Several options exist for specifying the node numbers at which the force acts, including:

- a single node;
- a range of nodes (first, last, increment); and
- all nodes on a plane defined by Ax + By + Cz = D.

The vector force (Fx, Fy, Fz) input is assumed to act at each node selected by one of the above methods. It should also be noted that the 'load case' number, a term usually associated with linear finite element analysis, can be used to distinguish sets of loads which act non-proportionally in a nonlinear solution (see REFMT, Subsection 5.1).

Element loads may be specified for a single element at a time, a range of elements (defined by first and last element numbers and an element number increment), or all elements in the model. For each series of elements selected, a single load specification can be defined by a loading case number, a loading type, and a load magnitude. Loading types are defined as follows:

- -1, -2, -3 : denotes a body force (force per unit volume), acting in the global X, Y, Z directions, respectively;

- 1,2,3,4,5,6 : for 3-D elements, signifies a distributed surface pressure acting on the surface (-R,-S,-T,+R,+S,+T), respectively;

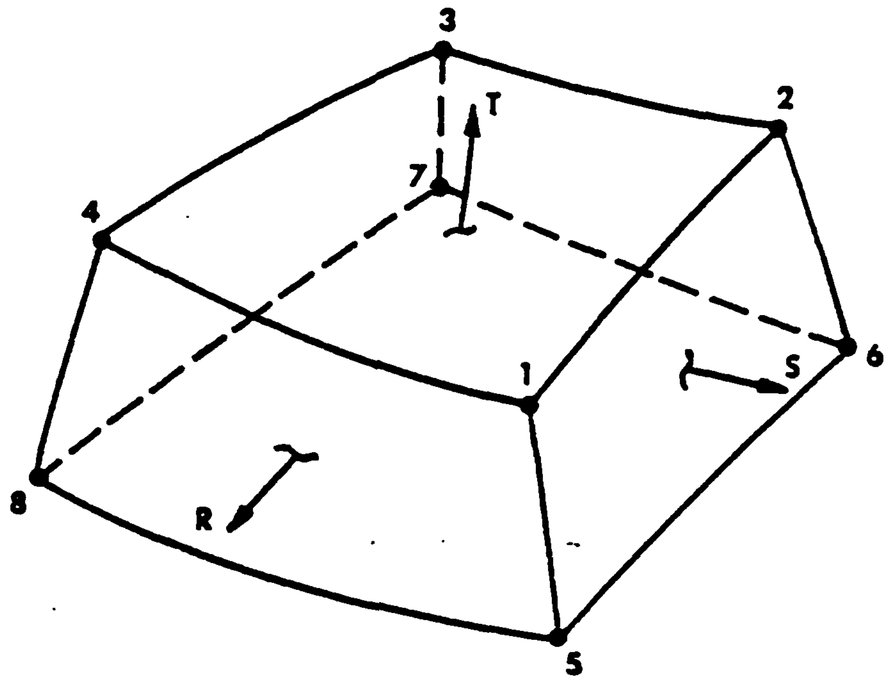
- 1,2,3,4 : for 2-D elements, describes an edge force (force per unit length) on edges 1, 2, 3 or 4, respectively; and

- 1,2,3,4 : for axisymmetric elements, defines a surface pressure on edge 1, 2, 3 or 4 of the element.

Surface numbers (for pressure loading on 3-D elements) and edge numbers (for line loads or axisymmetric pressures) are shown in Figures 4.6 and 4.7. The correct surface or edge number can be identified by recalling the numbering sequence (-R, -S, -T, +R, +S, +T for 3-D, and -R, -S, +R, +S for 2-D), and using the PLOT option ORIENT (see Subsection 4.6) to display the local coordinate directions for the elements in question.

4.5 NODE POINT REORDERING

Finite element models generated using interactive modelling techniques are typically completed with most of the nodal and element data being numbered in an almost random fashion. For efficiency in the actual finite element solution, reordering of either node points or elements is usually advisable, since the number of operations performed in the solution, as well as the storage requirements, can be affected to a significant degree.



SURFACE	LOCATION	CORNER NODES
1	R = -1	2 - 3 - 7 - 6
2	S = -1	3 - 4 - 8 - 7
3	T = -1	5 - 6 - 7 - 8
4	R = +1	1 - 4 - 8 - 5
5	S = +1	1 - 5 - 6 - 2
6	T = +1	1 - 2 - 3 - 4

Figure 4.6. Surface Numbers for 3-D Pressure Loads.

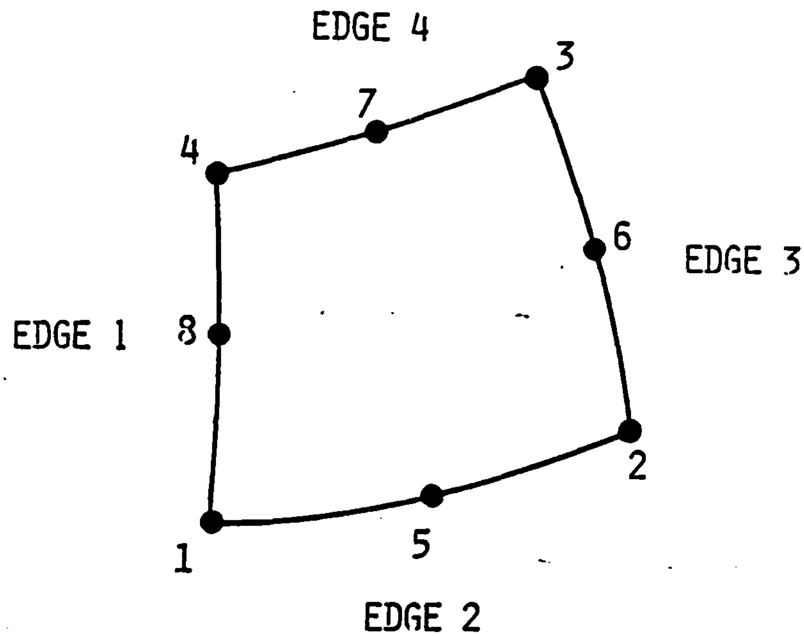


Figure 4.7. Edge Numbers for Line Loads for Axisymmetric Pressures.

An example of the effect of reordering a small finite element mesh upon the storage requirement for solution is depicted in Figure 4.8.

Node point reordering is performed in PREP using the RENUMBER command, which requires no input from the user other than an input file label. The RENUMBER option performs a renumbering of the nodes which is not optimum, but which is usually sufficient to reduce storage and solution time requirements to a very reasonable level. In particular, the renumbering scheme is generally quite effective when the equation solver to be used employs a variable bandwidth (profile, or envelope) storage mode [4].

RENUMBER displays the initial and final 'nodal bandwidths' as a measure of how effective the node reordering has been. The nodal bandwidth is the maximum bandwidth of a system of equations, based on the mesh in question, having a single degree of freedom per node; the actual maximum bandwidth for the model is the nodal bandwidth multiplied by the number of freedoms per node (usually 2, 3 or 6).

The RENUMBER utility is quite fast in execution, and can be executed more than once to attempt further reduction in the system bandwidth. Two passes through RENUMBER are recommended, to verify that the model bandwidth has been reduced as much as possible.

4.6 PLOTTING UTILITIES

Geometry plotting may be performed in PREP at any time by entering the PLOT command. PREP requests the name of an active data file to be plotted, and enters the plot command mode (identified by the prompt symbol "*").

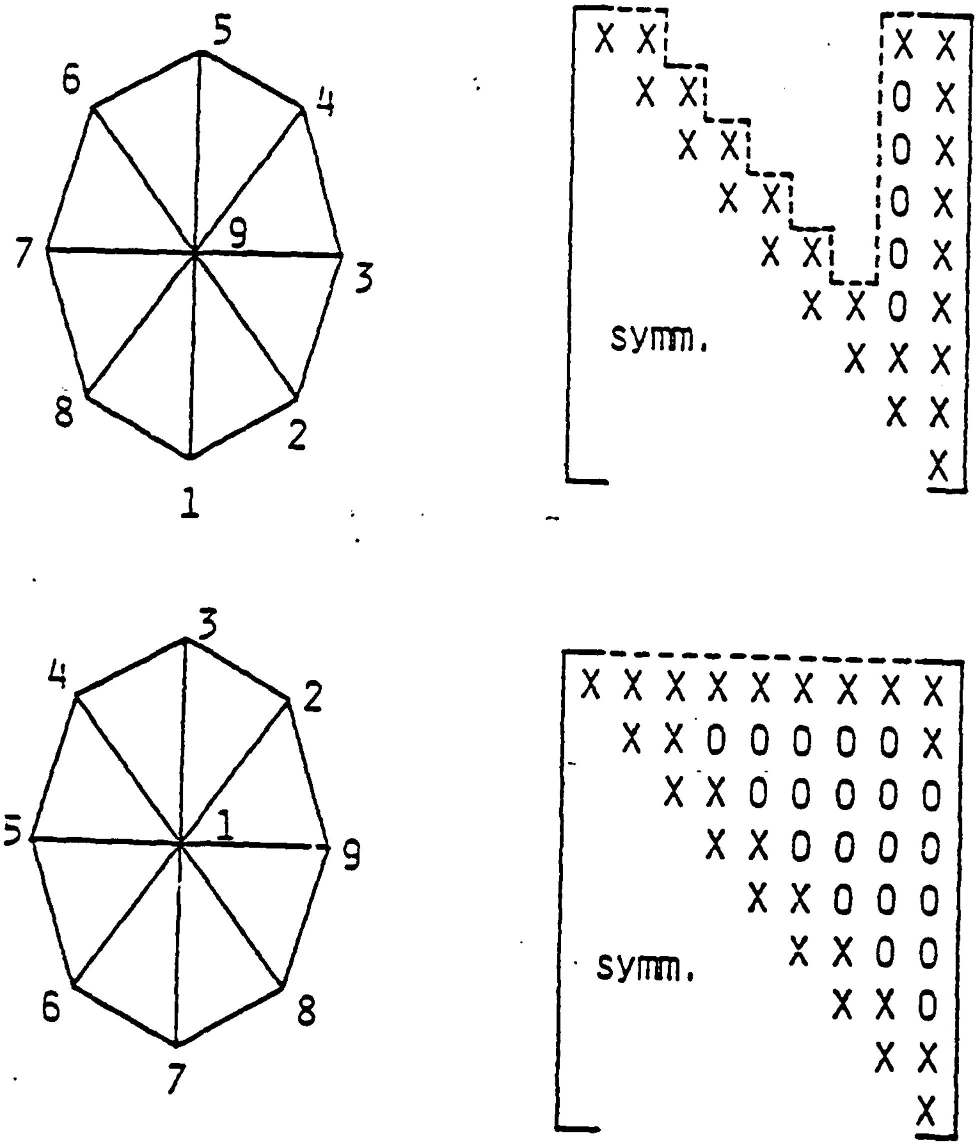


Figure 4.6 Example of Node Point Numbering Optimization.

In the PLOT mode, plotting commands may be entered to control the appearance of plots to be made. For example, the EYE command sets up a viewer position for use in plotting, but the plot is not actually drawn until a DRAW command is issued. Most of the PLOT mode commands control one or more plotting parameters, which may be set and reset as often as desired. These parameter-related commands include:

- AXES : control plotting of X, Y, Z axis lines
- CLIP : define a "clipping plane" which limits drawn lines to a specified minimum distance from the viewer
- CUBE : define minimum and maximum plotting coordinates
- DEFAULT : restore all plotting parameters to default values
- ELEMENT : select a range of elements to be plotted
- EYE : define the viewer eye position
- LABEL : select node and/or element numbering
- ORIENT : control plotting of local element orientations needed in REFINE and LOADS operations
- PROJECT : select orthogonal or perspective projection
- REFLECT : select reflection of geometry about a certain plane
- ROTATE : define rotations (for plotting purposes only)
- SCALE : scale or unscale plot
- SHRINK : select exploded view to be plotted
- TRANSLATE: define translations (for plotting only)
- VERTICAL : select a vertical axis direction
- ZOOM : select an area of the screen for close-up views

Sufficient default values are set upon entry to PLOT that the model may be drawn without setting any of the above parameters, and often this is a useful procedure for "getting oriented" with a new model. Once any of the above parameters has been set, however, the new plotting parameters will remain active even when PLOT is terminated and reentered; this measure eliminates the need for entering the same sequence of commands every time PLOT is executed.

The remaining commands available in the PLOT module are control functions, namely:

- BOUNDS : Plot symbols indicated boundary conditions at nodes
- DRAW : Plot the model using all current parameter values
- HELP : Print a list of available commands and options
- MAIN : Return control to PREP following plotting
- SITE : Redefine center of plotting region
- SUMMARY : Print a summary of all plotting parameter values
- TIME : Print the CPU time used since sign-on

All of the control functions request some form of immediate action, with no further input required from the user.

Most of the plotting options described above are demonstrated in the modelling examples in Section 7.

4.7 DATA LISTING UTILITIES

The functions available for listing of data within PREP are of two types. The LIST command produces a table of active model files, with their corresponding file numbers. The PRINT utility allows data from a specific model file to be printed at the terminal or on a formatted file suitable for line printer output. A typical LIST output might appear as follows:

```
LISTING OF DATA FILES AVAILABLE

#10 --- PANEL
#11 --- PANELREF
#13 --- PANEL + BEAM
```

Here the names 'PANEL', 'PANELREF', and 'PANEL + BEAM' are all names of active data files which may be specified as input for any PREP operation. The corresponding file numbers (10, 11, 13) are important only on CDC computers, where they correspond to the local file names TAPE10, TAPE11 and TAPE13, respectively. Upon leaving PREP, the model 'PANEL + BEAM' would be saved by cataloging the local file TAPE13 as a permanent disk file, for example. In the VAX 11/780 version of PREP, data files may be saved by name prior to leaving the program.

With the PRINT command, the contents of a single model may be listed at the terminal and/or written to a formatted file for off-line printing. Printing of header data, nodal coordinates, element data, boundary conditions, and nodal or element loads may each be selected as needed. For the nodal and element data, which can become quite lengthy, ranges of nodes or elements may be specified for printing.

4.8 TYPICAL SEQUENCES OF OPERATIONS

A complete listing of the commands and options available in PREP appears in Table 4.8. These commands can be grouped logically into four categories,

- utility commands,
- geometric modelling functions,
- boundary conditions and loads, and
- property definition.

The utility commands are those which perform data management or informative functions; generally, they may be executed at any time, in any sequence. The commands which fall into this category include:

TABLE 4.8 - Alphabetized Listing of PREP Commands

COMMAND	DESCRIPTION
BOUNDS	Specify nodal constraints
CONTACT	Create contact elements from surfaces of 3-D solids
COPY	Make a copy of a specified model file
CREATE	Begin a new model file by keyboard entry of data
DELETE	Delete a model file
EDIT	Make minor editing changes to an existing model
FILL	Generate midside and/or interior nodes
HELP	List available commands and options
LINEAR	Generate linear constraint conditions at nodes
LIST	List all active model files and file numbers
LOADS	Define nodal forces and/or distributed element loads
MASK	Eliminate midside and/or interior nodes
MERGE	Combine two model files
NAME	Assign a descriptive name to a file number
PLOT	Enter plotting mode
PRINT	Print all or part(s) of a model data file
PROPERTY	Assign element property data
REFINE	Subdivide elements to create a finer mesh
REFLECT	Form the mirror image of an entire model
RENUMBER	Reorder node points to reduce solution time
ROTATE	Rotate model geometry in space
SHELL	Select certain elements to be thin shells
SIFT	Eliminate unused node points
STOP	Exit from PREP
TRANSLATE	Translate model geometry in space
TIDY	Eliminate duplicate node points
TIME	Print CPU time since start of session
TOLERANCE	Reset tolerance for detecting coincident nodes

COPY	NAME
DELETE	PLOT
HELP	PRINT
LIST	TIME

Property definiton functions, which are also relatively benign with respect to their execution sequence, include

CONTACT
PROPERTY
SHELL

However, it may be advantageous in some applications to segregate elements of one type or property set from those of another early in the model generation, when fewer elements are involved. Definition of many different sets of properties in a complicated model presents a potential source of errors and oversights.

With the remaining two groups of commands, experience has shown that some sequences of commands result in very fast and effective modelling, while others can lead to long and tedious sessions at the terminal. It is generally recommended that the definition of loads and boundary condition data be performed as the LAST step in model generation, after such operations as REFINE, RENUMBER, etc. The geometric modelling operations also have a preferred order, which can best be described as three 'stages' of the geometric model generation. Commands associated with these modelling stages are summarized below:

Stage 1 :	FILL	MASK	
Stage 2 :	MERGE	REFINE	REFLECT
	ROTATE	TOLERANCE	TRANSLATE

Stage 3 : RENUMBER

In Stage 1, irregular elements entered in CREATE, and/or higher order elements which are to be reduced to lower-degree nodal patterns are adjusted to contain the final number of nodes per element which is to be used throughout the mesh. Once these steps have been accomplished, other operations such as REFINE and MERGE should proceed smoothly.

In Stage 2, the actual mesh division to be used for analysis is created, by REFIning a coarse mesh, performing TRANslations and ROTAtions to reproduce periodic sections of the grid, and MERGing disjoint sections of the model together. Since most of the operations in this phase of the modelling will affect the number and/or ordering of nodal coordinates and elements, RENUmbering, BOUNdary conditions and LOADs are best left until this stage is complete.

In Stage 3, the geometry of the mesh (number of nodes and elements and their relationship to one another) is complete, and RENUmber can be used to minimize the effects of the almost random numbering of the finite element model.

As the final step in generating the model, boundary conditions and loading data are entered in PREP. At this point, the geometry of the model, as well as the numbering of nodes and elements, is fixed; PLOT should be used to obtain the node and element numbers needed for specifying boundary conditions and loading.

SECTION 5

MODEL DATA TRANSLATION

The preprocessor contains two facilities for producing formatted output from PREP-generated finite element models. These are the REFMT and NEUTRAL utilities, which are described in Subsections 5.1 and 5.2, respectively.

REFMT provides output in a format compatible with the MAGNA nonlinear finite element program [4]. Additional data required by MAGNA which is not prepared in PREP can also be generated in REFMT interactively. In most instances, the REFMT-generated data deck is ready for submission directly to MAGNA for analysis. When special options or minor data changes are necessary, the formatted data file can be modified 'by hand' using the resident computer's editing facilities.

The NEUTRAL utility translates the PREP-format data file into a formatted form which is listable on the line printer and may be transferred between computer systems with a minimum of difficulty. NEUTRAL also performs the inverse translation function (back to PREP internal format) needed to re-introduce the finite element data into the preprocessor on another computer.

5.1 REFMT - MAGNA Data Translation

The REFMT translator accepts as input the PREP internal format data file, and generates a formatted data deck suitable for use with the finite element analysis program MAGNA [4]. All of the finite element types, and most of the special options, available in MAGNA are supported by REFMT.

In addition to reformatting the preprocessor-generated data, REFMT requests information from the analyst to complete the solution data needed in MAGNA. This additional information includes:

- the type of solution (linear or nonlinear, static or dynamic),
- information concerning restart files for nonlinear analysis,
- postprocessing file options, and
- nonlinear solution strategy (iteration method, etc.).

The data required for multiple loading conditions, contact analysis and other options is generated automatically from the preprocessor data.

Additional editing of the model data is possible in REFMT for material properties and loading data. Material properties may be selected from a library of stored properties (see Appendix F), or entered directly at the keyboard. When the element properties are complete for all elements in the model, materials data is listed and can be modified selectively as the need arises.

Applied loading data may take on different meanings in linear and nonlinear analysis, and for transient dynamic and nonlinear solutions, the time history of the loading must also be specified. REFMT allows a moderate amount of editing of the loading data to suit the type of analysis to be performed. The options include:

- options to combine multiple load cases, or analyze separately;
- generation of time-history curves for use in nonlinear and/or dynamic analysis (several built-in curves may be selected and scaled as needed); and
- specification of non-proportional loading sets in nonlinear and transient analysis, with each set scaled independently.

Most of the options available within REFMT are demonstrated in the sample preprocessing sessions included in Section 7. It should also be noted that the MAGNA data deck (whether output from REFMT or hand-generated) may be translated back into the preprocessor file formats using the TRNSFR utility described in Subsection 3.1.

5.2 NEUTRAL - Neutral File Translation Program

The NEUTRAL utility performs two data translation functions:

- preprocessor internal format to neutral data file, and
- neutral data file to preprocessor internal format.

These two functions are inverses of one another, with all preprocessor data types supported.

The two data files (internal format and neutral format) are described in Subsections 6.1 and 6.2, respectively. The neutral file format exists solely for the purpose of storing the preprocessor data in a readable form which is easily transferred between computer systems. Thus, finite element data prepared using PREP on one system may be translated to the neutral format using NEUTRAL, transmitted to a completely different machine type, returned to the internal format using NEUTRAL, and used within PREP on the second machine. The translation to neutral format is also useful for archiving of data, either on punched cards or in text library format (using the LIBRARY utility on VAX 11/78Ø, or UPDATE on CDC machines).

Execution of NEUTRAL requires only the declaration of the files involved, and the selection of the type of translation to be performed. On CDC machines, the internal-format data file is local file UNFMT, and the formatted (neutral) file is the local file FMTDAT. With the VAX version of NEUTRAL, the necessary file names are requested as input at the beginning of execution.

SECTION 6

MODEL DATA FILES

Optimal data handling of information associated with model geometry and finite element analysis is highly dependent on how that data is accessed and stored. The preprocessing system utilizes several types and formats for data files to take advantage of specific features offered by each. Data that is transferred between the various preprocessing modules in the stages of model development, modification, and assignment of analysis parameters is stored in an internal, unformatted data file. This binary data file is only readable by programs that are properly equipped to read the particular file in question. Two different data files are employed in the internal, unformatted data type; one for the output data generated by CORGEN, SPATCH, or AGRID, and a second data type generated by PREP, IJKGEN, CREATE, or NEUTRAL. The second type of data file used in the preprocessor system is the external, formatted data file. This data file type is used for transferring model data between preprocessing programs, and for archival of data in a readable form. The external, formatted data file is a character text file and may therefore be read without any need for special programs. General descriptions of the form and purpose of each of these data file types are given in the remainder of this Section. Detailed descriptions of the file formats for each type are presented in Appendix A.

6.1 Preprocessor Internal Data Structure

The preprocessing system defines two functionally similar data structures for storing model geometry data. Execution of any of the data entry and interface programs discussed in Sections 2 and 3 results in the creation of either a two-dimensional surface model geometry file (see CORGEN, AGRID, and SPATCH) or a three-dimensional solid model geometry file (see CREATE, IJKGEN, NEUTRAL, TRNSFR, and IMPRINT). The surface model geometry file is an intermediate model file designed to be input to the EXPAND module to convert the nine node surface element model to a twenty-seven node solid element model. The shell surface file and the standard (PREP-format) 3-D geometry file are the subjects of Paragraphs 6.1.1 and 6.1.2, respectively.

6.1.1 Shell Surface Geometry File

Certain modelling requirements for windshield canopies and related structures fall into a class where the structures may be represented by 2-D quadratic surface elements located in 3-D space. If a thickness value is associated with the surface then it becomes a straightforward task to mathematically expand the surface to a 3-D solid representation. The CORGEN, AGRID and SPATCH programs generate simplified surface models which can be readily represented by the data file described in Appendix A, Table A.1. This data file contains coordinate data with an associated thickness value for each node, and element connectivity for 2-D bi-quadratic surface elements. The surface elements are defined by nine nodes: four corner nodes, four midside nodes and a center node. The expand module will convert the nine node surface element model to three-dimensional solid finite elements stored in the three dimensional model file format described below. This 2-D file is characterized by the implicit node numbering of coordinates and the implicit numbering of the elements.

6.1.2 Three-dimensional Model File

The requirements of maintaining data neccessary for model geometry as well as for numerical analysis, determine the structure of the three-dimensional model file. Table A.2 shows the details of this file structure. This model file is the workhorse of the preprocessing system in that all models that utilize the PREP and REFMT modules must be converted to this data structure. Particular features of this model file include the arrangement of element loads, nodal forces and boundary conditions in such a manner that this data need not be provided nor must file space be allocated to provide for it later. While this file type is similar to the surface model data structure the user should be aware of the explicit node and element numbering that is included with this data structure along with the loss of the thickness data required for EXPAND and the addition of element codes that regulate and affect the numerical solution options. This file is used as input to several of the preprocessing modules as illustrated in Figure 1.1. Once all preprocessing functions have been executed for model definition this file is input to the REFMT module to convert it to a reformatted character file with the correct options for input to the appropriate analysis program.

6.2 Neutral Data File Structure

The neutral data file is an external, formatted data file intended for use in archiving finite element data and transmitting model data from one computer system to another. The neutral data file is read and written only by the NEUTRAL utility (Subsection 5.2), which performs the translation between neutral files and the internal format of the preprocessor (Subsection 6.1).

The neutral file contains exactly the same information as the standard preprocessor data file (see Appendix A). However, the fact that the neutral file is a formatted (text) file has several useful consequences:

- data may be archived in neutral file form using the LIBRARY functions provided by VAX/VMS, or UPDATE on CDC machines;
- alternatively, model data may be punched directly on cards for later use; and
- transmission of data via RJE terminals can be accomplished with a minimum of difficulty.

The data sequence and formats for the neutral data file are listed in Appendix A of this report.

6.3 MAGNA Input Data File

The MAGNA [4] finite element analysis program requires the user to provide data for an analysis in a particular formatted form. While the data stored in the internal unformatted form is adequate for preprocessing functions, this data must be converted to the appropriate structure for input to MAGNA. The MAGNA input data file is created by the REFMT program in response to user selected solution options. This file contains the complete information concerning solution options, nodal coordinates, element connectivity, boundary conditions, loading requirements and all associated parameters for element types, material properties, time increments, etc. All data on this file is formatted according to the structure outlined in detail in Reference [4]. Due to the formatted nature of the MAGNA data file, the user may readily edit it directly to alter any values that may have been improperly specified in the REFMT program. In addition, the user is encouraged to make changes to the file directly, in the event minimal changes are necessary for altering solution options. Reference [4] will be necessary for the user wishing to alter data on this file.

SECTION 7

DEMONSTRATION PROBLEMS

Several modelling examples are presented in this Section which illustrate the capabilities of the preprocessor and the types of procedures involved in generating a finite element model. Three of the data entry programs (CREATE, CORGEN, and IJKGEN), which represent the most common points of entry into the system, are used for the creation of the original models. In the case of CORGEN, which generates a surface model file, the use of EXPAND to create a fully three-dimensional model is also demonstrated. In each example, further modelling operations are performed within PREP, and the final data is translated into the MAGNA input format using REFMT.

7.1 EXECUTION OF CREATE / PREP / REFMT SEQUENCE

The following example illustrates the generation of a two dimensional finite element model using the CREATE data entry program. The model is completed using PREP, and formatted data for analysis with MAGNA is generated using REFMT.

The model generated below involves a thin, flat rectangular plate with a circular hole at the center; only one quadrant of the structure is represented due to symmetry. Of particular interest is the technique of using only selected midside nodes for input in CREATE to capture the curvature of the boundary along the hole. This method allows a minimum of hand-generated input, but requires the use of FILL (or MASK) within PREP to generate a properly connected mesh of elements.

7.1.1 Summary of Modelling Procedure

The basic geometry of the plate is defined using CREATE, as detailed in the sample terminal session of Section 7.1.2. Ten node points and three elements are used to define the overall geometry. Note that only the corner nodes are used, except where curved boundaries are to be defined. It is important to observe that, if only selected midside nodes are used, the ordering of element connectivity may contain zeroes in positions corresponding to unused nodal positions. This situation occurs in the first two elements of the input mesh.

The primary operations performed in PREP are the FILL option, in which additional node points are generated to fill out the midside and element centroid positions, and a series of REFIne operations which subdivide each of the three major portions of the mesh to obtain a final model containing 80 nine-node elements.

The REFMT step specifies a linear, static analysis. Material properties, which were not defined within PREP, are also selected from the materials library during execution of REFMT. Following the execution of REFMT, the MAGNA input file (local file FDATA) is saved as the permanent file PLATEP on an indirect file library.

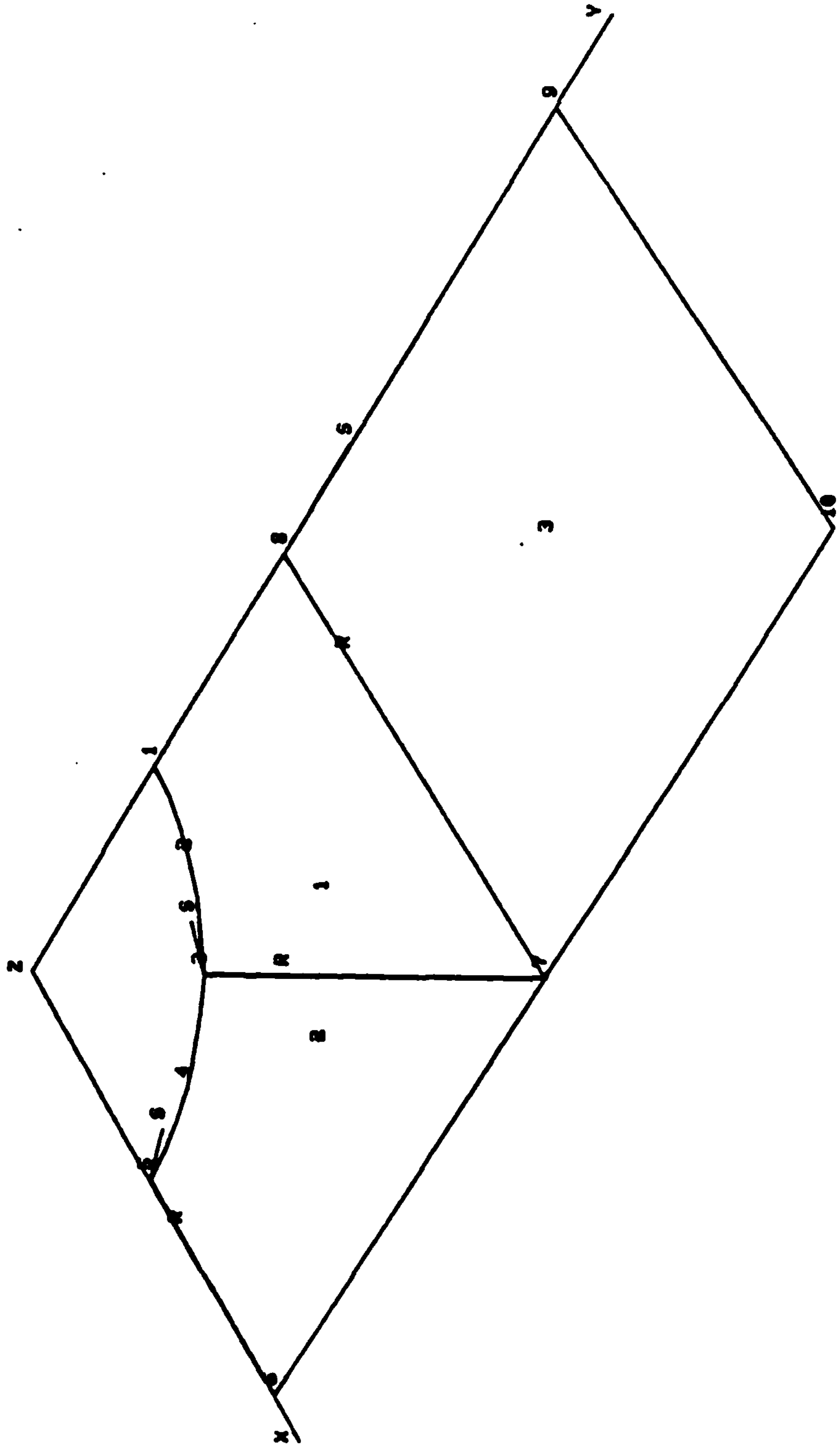
7.1.2 CREATE Execution

```
COMMAND- ATTACH,P,PREPROCESSORPROC,ID=BROCKMAN,MR=1.
 AT CY= 998 SN=AFFDL
COMMAND- BEGIN,CREATE,P.

  ***************************************
  BEGIN CREATE - DATA ENTRY AND EDITING OF
  COARSE GRID FINITE ELEMENT MODEL DATA
  ***************************************

  DO YOU HAVE A PREVIOUSLY GENERATED DATA
  FILE FROM CREATE TO RE-EDIT? (Y,N).....: N


  ***************-   INITIAL NODE POINT INPUT    ***************

  IN THIS PART OF THE INPUT, COORDINATE DATA IS ENTERED AT THE
  KEYBOARD FOR EACH NODE OF THE MODEL. THE DATA IS CHECKED FOR
  CONSISTENCY AND MAY BE EDITED LATER. THE MAXIMUM ALLOWABLE
  NODE POINT NUMBER IS 500.

  ENTER THE DIMENSIONALITY OF THE MESH TO
  BE DEFINED ( 2 OR 3 ) .................. 2

  *****-BEGIN DIRECT NODAL POINT INPUT -
  ENTER NODE NO. AND 2 COORDINATES AT EACH NODE
  ( ENTER ALL ZEROES TO TERMINATE INPUT )

  ENTER NODE, X, Y   - 1   0.     5.
  ENTER NODE, X, Y   - 2   1.87  4.6
  ENTER NODE, X, Y   - 3   3.55  3.55
  ENTER NODE, X, Y   - 4   4.6   1.87
  ENTER NODE, X, Y   - 5   5.    0.
  ENTER NODE, X, Y   - 6  10.    0.
  ENTER NODE, X, Y   - 7  10.   10.
  ENTER NODE, X, Y   - 8   0.   10.
  ENTER NODE, X, Y   - 9   0.   20.
  ENTER NODE, X, Y   -10  10.   20.
  ENTER NODE, X, Y   -0,0,0


  EDIT THE NODAL POINT DATA ( Y , N ).....N

  ****************    INITIAL ELEMENT INPUT    ****************

  THIS PORTION OF THE INPUT GIVES THE CONNECTION DATA DEFINING
  FINITE ELEMENTS IN TERMS OF THE NODES. CONNECTIVITY FOR THE
  ELEMENTS FOLLOWS THE CONVENTIONS FOR TWO AND THREE DIMENSIONAL
  ELEMENTS IN MAGNA ----------


  ****** TWO-DIMENSIONAL ******  ***** THREE-DIMENSIONAL *****
  NODE POINTS      LOCATIONS        NODE POINTS     LOCATIONS
     1 - 4         CORNERS             1 - 8        CORNERS
     5 - 8         MIDSIDES            9 - 20       MIDSIDES
       9           CENTROID           21 - 26       MIDFACES
                                         27         CENTROID

  THE MAXIMUM ALLOWABLE ELEMENT NUMBER IS 100.

  FOR EACH ELEMENT, ENTER (1)ELEMENT NUMBER, (2)NUMBER LOCAL
  NODES (I.E.,THE LENGTH OF THE CONNECTIVITY LIST) AND (3)THE
  LIST OF CONNECTED NODES.  ENTER ELEMENT=0 TO TERMINATE INPUT

  FOR 2-D ELEMENTS, THE MAX. LOCAL NODE NUMBER SHOULD BE BETWEEN  2 AND  9

  ENTER ELEMENT NO.-1
  MAX. LOCAL NODE  -6
  CONNECTIVITY LIST-8,1,3,7,0,2

  ENTER ELEMENT NO.-2
  MAX. LOCAL NODE  -6
  CONNECTIVITY LIST-7,3,5,6,0,4

  ENTER ELEMENT NO.-3
  MAX. LOCAL NODE  -4
  CONNECTIVITY LIST-10,9,8,7

  ENTER ELEMENT NO.-0
```

```
EDIT THE ELEMENT DATA ( Y , N )..........Y

ELEMENT EDITING OPTIONS - (L)IST, (I)NPUT, (D)ELETE, (E)XIT
ENTER EDITING OPTION ( L , I , D , E )--L
ENTER RANGE OF ELEMENTS TO BE LISTED ---1,3

ELMT     N1   N2   N3   N4   N5   N6   N7   N8   N9 N
  1       8    1    3    7    0    2
  2       7    3    5    6    0    4
  3      10    9    8    7

ELEMENT EDITING OPTIONS - (L)IST, (I)NPUT, (D)ELETE, (E)XIT
ENTER EDITING OPTION ( L , I , D , E )--E


* SUMMARY OF CURRENT MODEL PARAMETERS *

HIGHEST-NUMBERED NODE POINT.........   10
NUMBER OF UNDEFINED NODES...........    0
HIGHEST-NUMBERED ELEMENT DEFINED....    3
NUMBER OF UNDEFINED ELEMENTS........    0

NUMBER OF REFERENCES TO UNDEFINED NODES=    0

EDIT THE NODAL POINT DATA ( Y , N )......N

EDIT THE ELEMENT DATA ( Y , N )..........N

****************************************
 PREPROCESSOR DATA FILE COMPLETE
****************************************

    STOP
    041400 MAXIMUM EXECUTION FL.
     0.168 CP SECONDS EXECUTION TIME.
COMMAND- FILES
--LOCAL FILES--
 ZP       SINPUT     SOUTPUT    UNFMT
COMMAND-
```

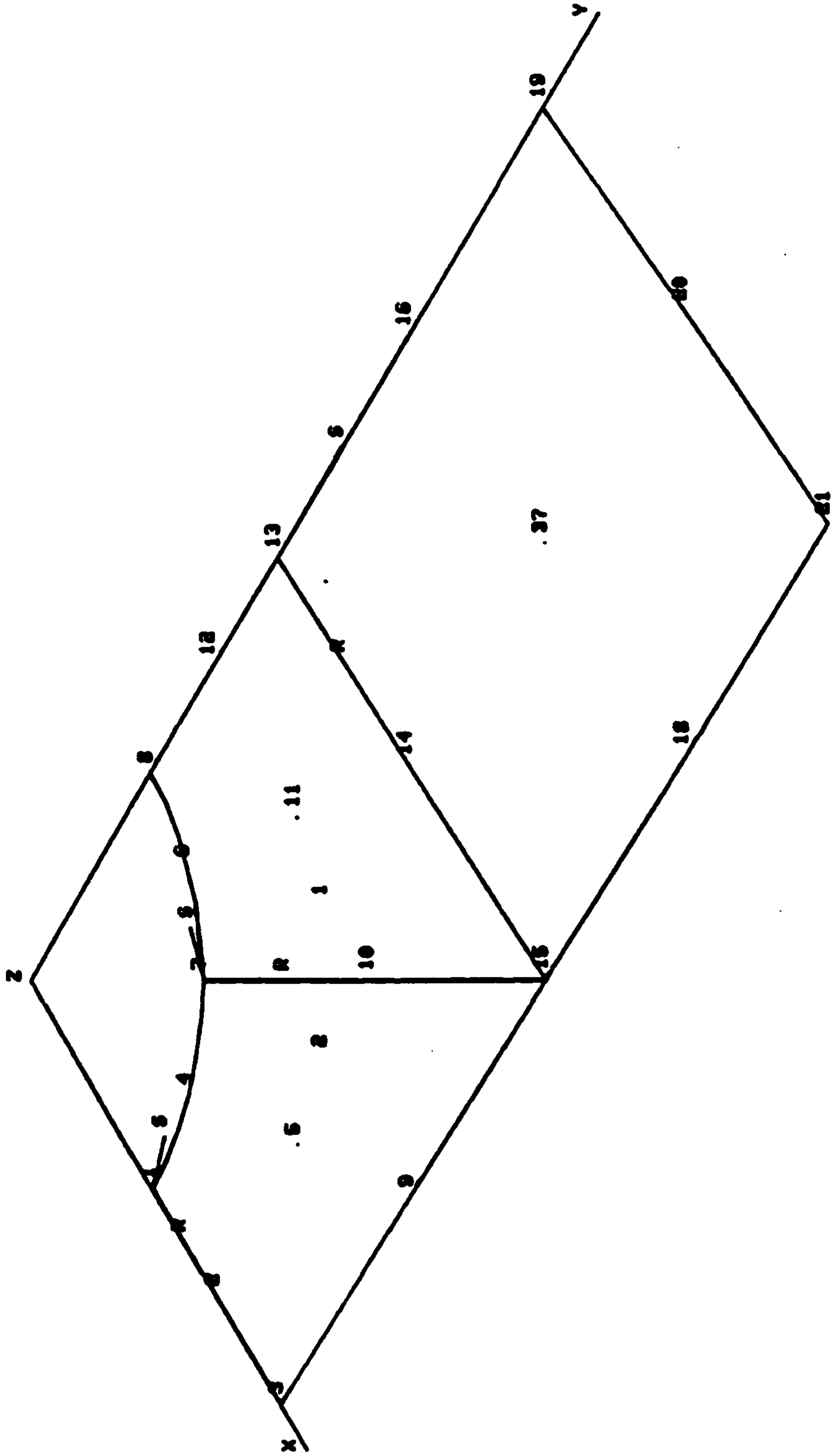
7.1.3 PREP Execution

```
COMMAND  FILES
--LOCAL FILES--
 *P          $INPUT      $OUTPUT      UNFMT
COMMAND- REWIND,UNFMT
COMMAND- COPYBF,UNFMT,TAPE10
 EOI ENCOUNTERED AFTER COPY OF FILE
         0, RECORD  1
COMMAND- RETURN,UNFMT
COMMAND- BEGIN,PREP,P.

          ****************************************
          *************  BEGIN PREP  *************
          **** MAGNA PREPROCESSING UTILITIES *****
          ****************************************


PLOTTING TERMINAL ( Y , N ) ?;;Y
TEKTRONIX OR HP ( T , H ) ?  ;;T

TEKTRONIX TERMINAL TYPES ----
 0, 4006-1
 1, 4010 / 4012 / 4013 / 4052
 2, 4014 / 4015
 3, 4014 / 4015   ( ENH. GR. MOD. )
 4, 4114

ENTER TERMINAL TYPE ( 0 - 4 );;3
ENTER CHARACTERS PER SECOND  ;;120
NUMBER OF INPUT DATA FILES ? ;;1
ENTER FILE #                 ;;10
ENTER LABEL                  ;;PLATE

TYPE 'HELP' FOR LIST OF COMMANDS

;; PLOT
ENTER INPUT FILE #1 LABEL              ;;PLATE

INITIALIZATION OF PLOTTING ROUTINE ERASES SCREEN.
READY (Y,N) ? ....: Y


:HELP
  COMMAND          DESCRIPTION
  *******          ***********
   AXES            AXES DRAW AND LABEL
   BOUNDS          PLOT BOUNDARY CONDITIONS
   CLIP            CLIP PLANE POSITION
   CUBE            SET MINIMA AND MAXIMA
   DEFAULT         SET DEFAULT VALUES
   DRAW            DRAW MODEL
   ELEMENTS        PLOTS ALL OR SELECTED ELEMENTS
   EYE             EYE POSITION
   HELP            LIST ALL COMMANDS
   LABELS          LABELS ELEMENTS AND/OR NODES
   MAIN            RETURNS CONTROL TO MAIN PROGRAM
   ORIENT          PLOTS R,S,T ORIENTATION AXES
   PROJECTION      PROJECTION TYPE
   REFLECT         REFLECT A PLANE
   ROTATE          ROTATE MODEL ABOUT AXES
   SCALE           SCALE PLOT
   SHRINK          SHRINK ELEMENTS
   SUMMARY         LIST ALL PARAMETER VALUES
   TIME            PRINT CPU TIME SINCE START OF SESSION
   TRANSLATE       TRANSLATE MODEL FROM ORIGIN
   VERTICAL        VERTICAL AXIS
   ZOOM            ZOOM ON THE MODEL
   SITE            SITE POSITION
:AXES
  PLOT AND LABEL THE AXES?(Y,N) ...............: Y
:LABE
   LABEL THE ELEMENTS?(Y,N) ....................: Y
   LABEL THE NODES?(Y,N) .......................: Y
   LABEL ALL THE SURFACES?(Y,N) ................: Y
:ORIENT
   PLOT ORIENTATION AXES?(Y,N) .................: Y
:DRAW

 AT END OF PLOT, ENTER CHARACTER TO CONTINUE.
 READY (Y,N) ? ....: Y
```

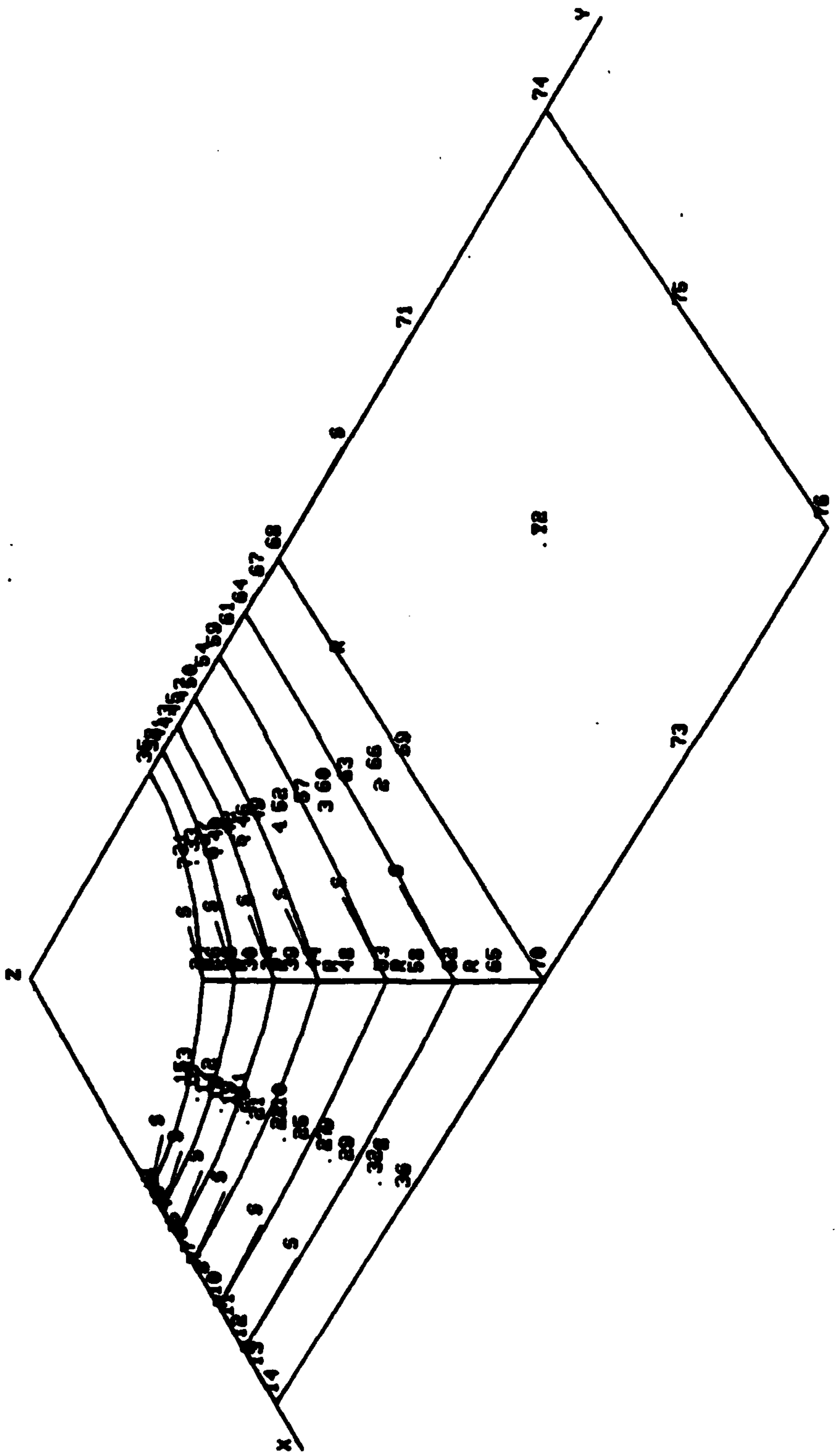


7.7

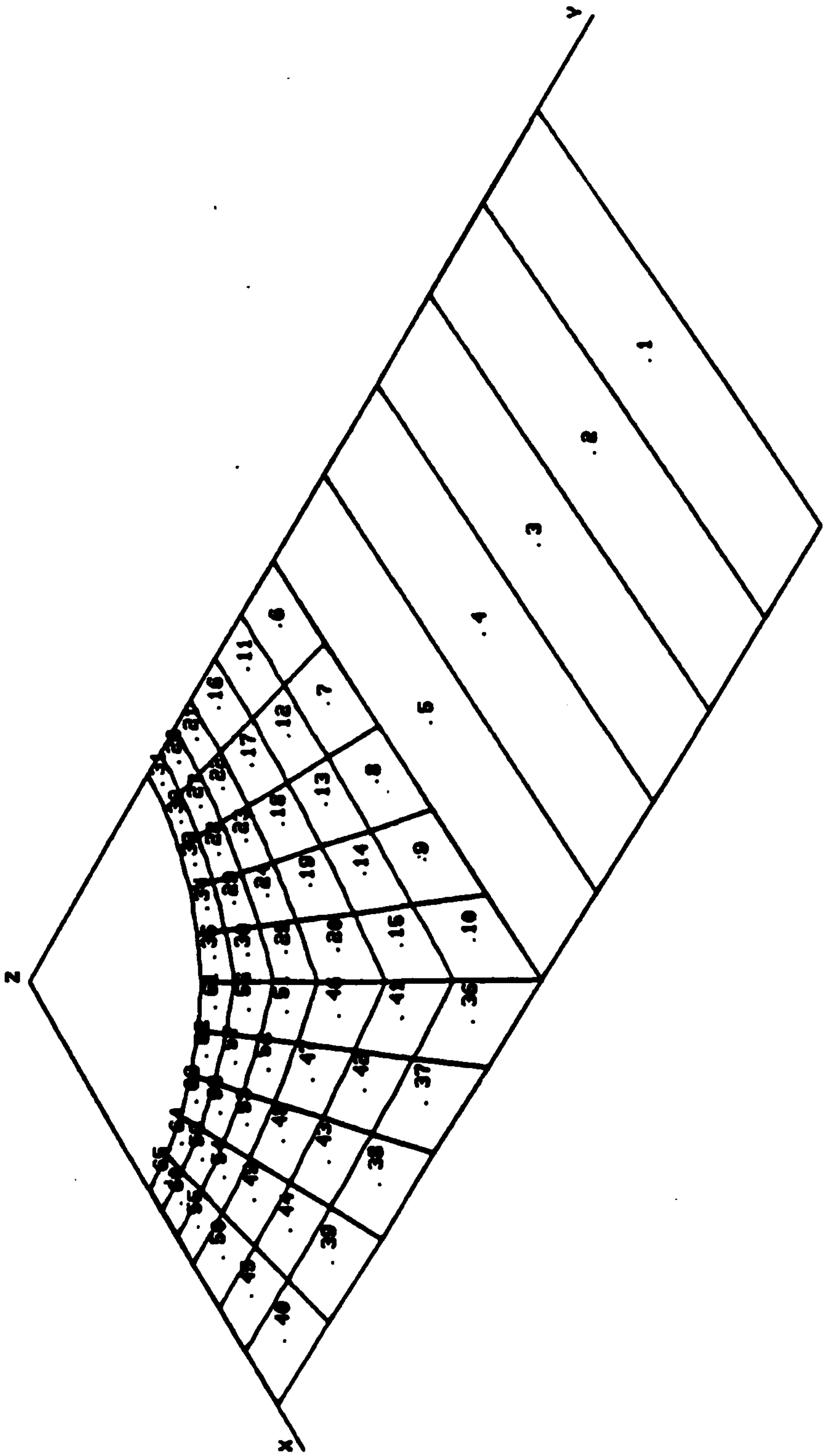
```
 *MAIN

:: FILL
ENTER INPUT FILE #1 LABEL                    ;;PLATE
ENTER OUTPUT FILE LABEL                      ;;PLAT2
CREATING NEW DATA FILE
FILL OPTIONS -
  1. 3-D ELEMENTS ONLY
  2. 2-D ELEMENTS ONLY
ENTER FILL OPTION (1,2)....;;2
MAX NODE POSITION # IN FINAL ELEMENTS?
  (8,9)...;;9

FILL COMPLETE.
PERFORM SUBSEQUENT TIDY ? (Y,N) ->Y

TIDY OPTION ONE OR TWO ? (OPT1, OPT2, HELP) ->OPT2

      TIDY COMPLETE
    2 DUPLICATE NODES DELETED

:: PLOT
ENTER INPUT FILE #1 LABEL                    ;;PLAT2
 *DRAW

  AT END OF PLOT, ENTER CHARACTER TO CONTINUE.
  READY (Y,N) ? ....: Y
```


X
Y
Z
R
19
16
13
12
8
14
10
18
9
.37
.11
.5

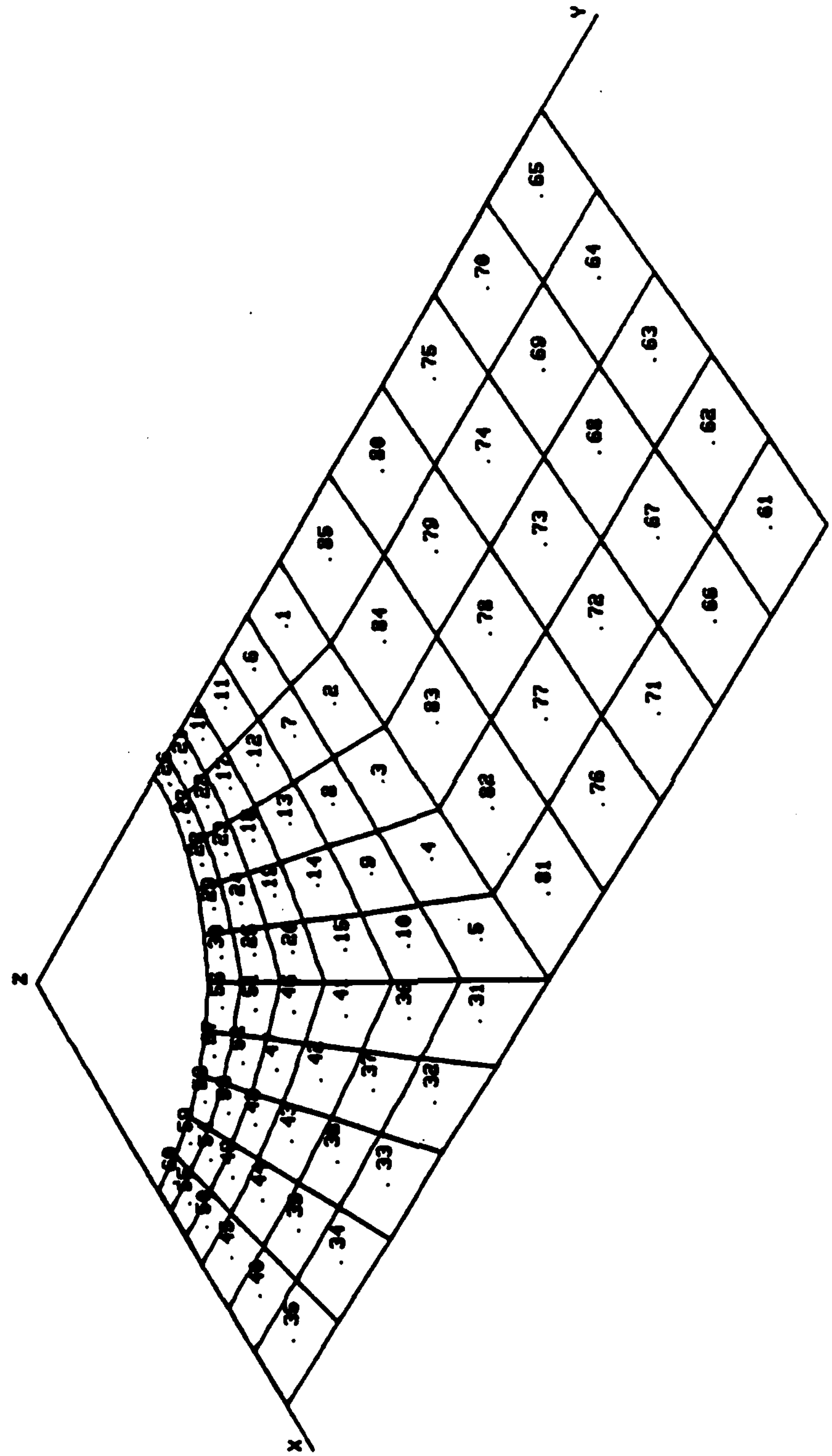
```
 !! :I

:: .I:T

LISTING OF DATA FILES AVAILABLE

#10 --- PLATE
#11 --- PLAT2

:: DELE
ENTER LABEL                    ;;PLATE

:: REFINE
ENTER INPUT FILE #1 LABEL                ;;PLAT2
ENTER OUTPUT FILE LABEL                  ;;REFIN1
CREATING NEW DATA FILE

  BEGIN REFINE ROUTINE...

1 - RANDOM ELEMENT INPUT
2 - RANGE OF ELEMENTS
3 - EXECUTE REFINEMENT
ENTER OPTION --------------->1
MAXIMUM NUMBER OF ELEMENTS
  SPECIFIABLE -------------->  150
NUMBER STILL AVAILABLE ----->  150

ENTER ELEMENT NUMBER AND <RETURN> FOR EACH
  ELEMENT TO BE REFINED.  ELEMENTS NEED NOT BE
  ENTERED IN ASCENDING ORDER.  ELEMENT NUMBER 0
  TERMINATES INPUT.
ENTER ELEMENT # ------------>1
ENTER ELEMENT # ------------>2
ENTER ELEMENT # ------------>0

1 - RANDOM ELEMENT INPUT
2 - RANGE OF ELEMENTS
3 - EXECUTE REFINEMENT
ENTER OPTION --------------->3

ENTER CUT DIRECTION (R,S,T) >R

MAXIMUM CUTS PER ELEMENT ---> 5
ENTER CUTS PER ELEMENT ----->5

EQUALLY SPACED CUTS ? (Y,N) >N

ENTER CUT POSITIONING IN
  ASCENDING ORDER (0-100) -->10 22 36 55 75

REFINE COMPLETE.
PERFORM SUBSEQUENT TIDY ? (Y,N) ->Y

TIDY OPTION ONE OR TWO ? (OPT1, OPT2, HELP) ->OPT2


     TIDY COMPLETE
   11 DUPLICATE NODES DELETED

:: PLOT
ENTER INPUT FILE #1 LABEL                ;;REFIN1
 #DRAW

  AT END OF PLOT, ENTER CHARACTER TO CONTINUE.
  READY (Y,N) ? ....: Y
```



```
% RUN

:: REFINE
ENTER INPUT FILE #1 LABEL                    ;;REFIN1
ENTER OUTPUT FILE LABEL                      ;;REFIN2
CREATING NEW DATA FILE

  BEGIN REFINE ROUTINE...

1 - RANDOM ELEMENT INPUT
2 - RANGE OF ELEMENTS
3 - EXECUTE REFINEMENT
ENTER OPTION --------------->2
MAXIMUM NUMBER OF ELEMENTS
  SPECIFIABLE -------------->  150
NUMBER STILL AVAILABLE ----->  150

ENTER BEGINNING ELEMENT NUMBER, ENDING ELEMENT
  NUMBER, AND INCREMENT FOR RANGE.
ENTER NBEG,NEND,INCR ------->1,13,1

1 - RANDOM ELEMENT INPUT
2 - RANGE OF ELEMENTS
3 - EXECUTE REFINEMENT
ENTER OPTION --------------->3

ENTER CUT DIRECTION (R,S,T) >S

MAXIMUM CUTS PER ELEMENT ---> 5
ENTER CUTS PER ELEMENT ----->4

EQUALLY SPACED CUTS ? (Y,N) >Y

REFINE COMPLETE.
PERFORM SUBSEQUENT TIDY ? (Y,N) ->Y

TIDY OPTION ONE OR TWO ? (OPT1, OPT2, HELP) ->OPT2

      TIDY COMPLETE
  119 DUPLICATE NODES DELETED

:: PLOT
ENTER INPUT FILE #1 LABEL                    ;;REFIN2
 %ORIENT
  PLOT ORIENTATION AXES?(Y,N) ..................: N
 %LABEL
  LABEL THE ELEMENTS?(Y,N) .....................: Y
  LABEL THE NODES?(Y,N) ........................: N
 %DRAW

  AT END OF PLOT, ENTER CHARACTER TO CONTINUE.
  READY (Y,N) ? ....: Y
```


Y
Z
X

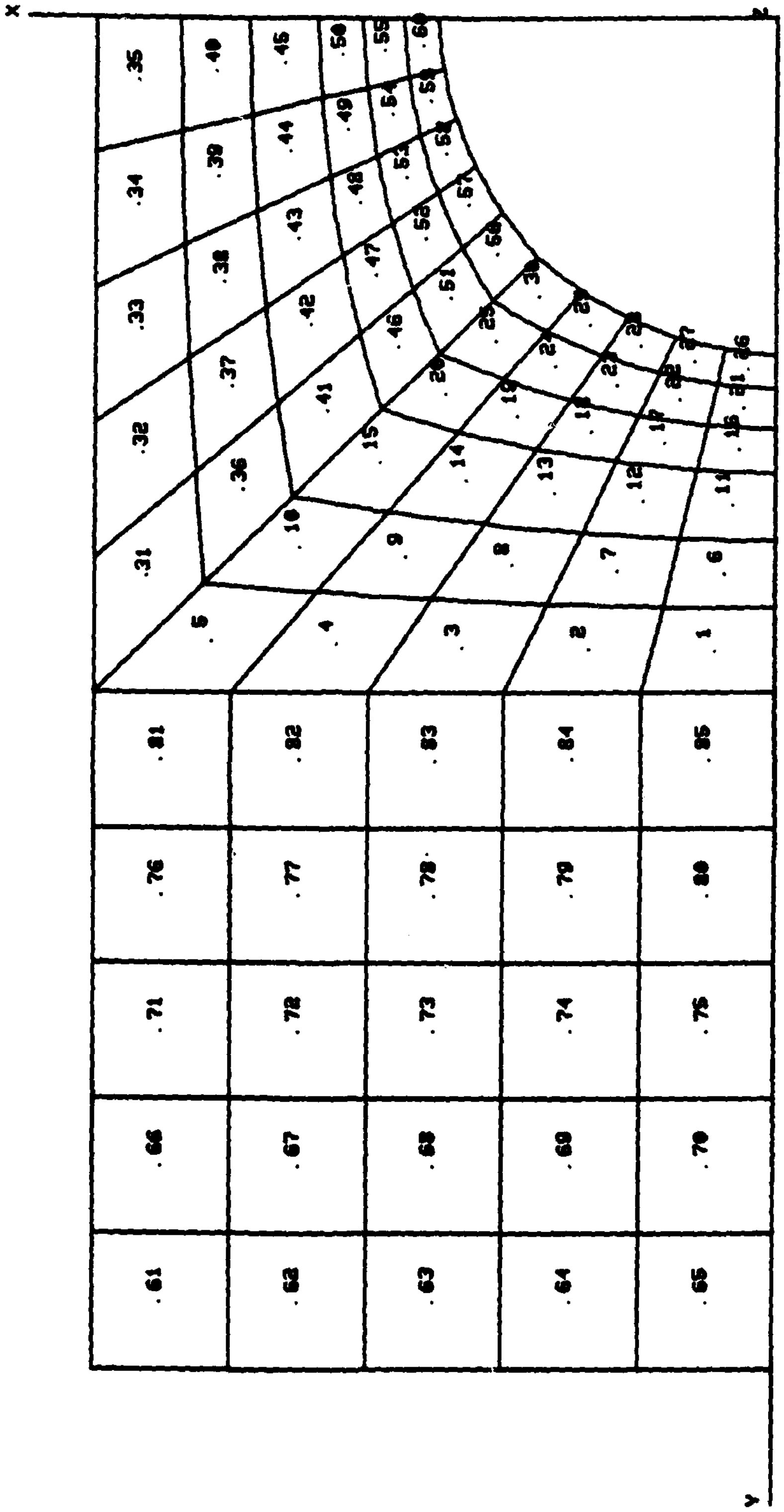
```
MAIN

:: LIST

LISTING OF DATA FILES AVAILABLE

$10 --- REFIN1
$11 --- PLAT2
$12 --- REFIN2

:: DELE
ENTER LABEL                    ;;REFIN1

:: DELE
ENTER LABEL                    ;;PLAT2

:: REFINE
ENTER INPUT FILE $1 LABEL                ;;REFIN2
ENTER OUTPUT FILE LABEL                  ;;REFIN3
CREATING NEW DATA FILE

  BEGIN REFINE ROUTINE...

1 - RANDOM ELEMENT INPUT
2 - RANGE OF ELEMENTS
3 - EXECUTE REFINEMENT
ENTER OPTION -------------->2
MAXIMUM NUMBER OF ELEMENTS
  SPECIFIABLE ------------->  150
NUMBER STILL AVAILABLE ----->  150

ENTER BEGINNING ELEMENT NUMBER, ENDING ELEMENT
  NUMBER, AND INCREMENT FOR RANGE.
ENTER NBEG,NEND,INCR ------->1,5,1

1 - RANDOM ELEMENT INPUT
2 - RANGE OF ELEMENTS
3 - EXECUTE REFINEMENT
ENTER OPTION -------------->3

ENTER CUT DIRECTION (R,S,T) >R

MAXIMUM CUTS PER ELEMENT ---> 5
ENTER CUTS PER ELEMENT ----->4

EQUALLY SPACED CUTS ? (Y,N) >Y

REFINE COMPLETE.
PERFORM SUBSEQUENT TIDY ? (Y,N) ->Y

TIDY OPTION ONE OR TWO ? (OPT1, OPT2, HELP) ->OPT2

     TIDY COMPLETE
   55 DUPLICATE NODES DELETED

:: PLOT
ENTER INPUT FILE $1 LABEL                ;;REFIN3
 XDRA
   INVALID COMMAND, ENTER 'HELP' FOR LIST OF COMMANDS
 XDRAW

  AT END OF PLOT, ENTER CHARACTER TO CONTINUE.
  READY (Y, ) ? ....! Y
```



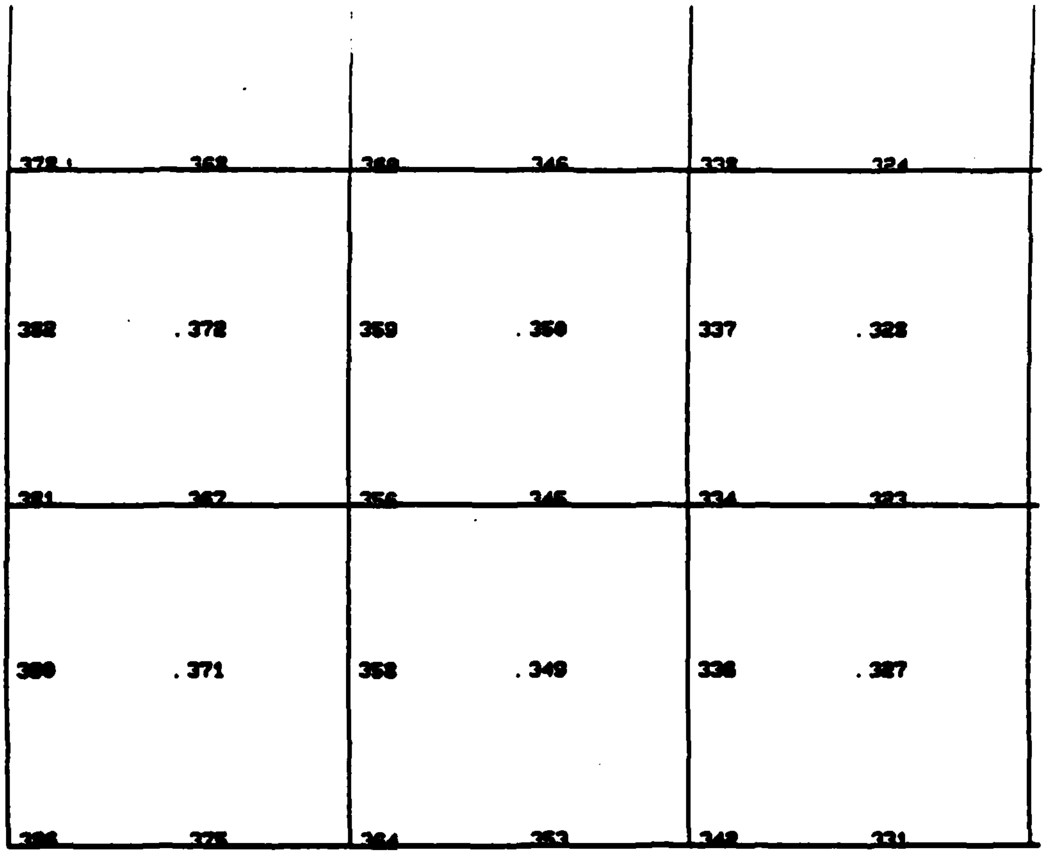
y
z
x

```
:: LIST

LISTING OF DATA FILES AVAILABLE

#10 --- REFIN3
#12 --- REFIN2

:: DELE
ENTER LABEL                    ;;REFIN2

:: RENUM
ENTER INPUT FILE #1 LABEL                ;;REFIN3
ENTER OUTPUT FILE LABEL                  ;;PLATE-4
CREATING NEW DATA FILE

 NODE POINT REORDERING COMPLETE

 NODAL BANDWIDTH (OLD) =   97
 NODAL BANDWIDTH (NEW) =   97

:: DELE
ENTER LABEL                    ;;REFIN3

:: LIST

LISTING OF DATA FILES AVAILABLE

#11 --- PLATE-4

:: BOUNDS
ENTER INPUT FILE #1 LABEL                ;;PLATE-4
ENTER OUTPUT FILE LABEL                  ;;PLATE+BCS
CREATING NEW DATA FILE

BOUNDARY CONDITIONS  MAY  BE ENTERED  FOR  RANDOM
  NODES,  RANGES OF NODES,  ALL NODES ON A PLANE,
  OR THE ENTIRE MODEL.

FOR EACH GROUP OF NODES,  CONSTRAINTS  MAY BE SET
  IN ANY COMBINATION OF THE X, Y, & Z DIRECTIONS.
  (NOTE THAT 2-D MODELS MUST BE FIXED IN Z)

CONSTRAINT DIRECTION CODE CONSTRUCTED AS FOLLOWS:

  ENTRY 1 - '1' FOR CONSTRAINED IN X DIRECTION
            '0' FOR UNCONSTRAINED IN X DIRECTION
  ENTRY 2 -     SAME FOR Y
  ENTRY 3 -     SAME FOR Z

  EXAMPLE -> '1,0,1' FOR CONSTRAINTS IN X AND Z
                     DIRECTIONS BUT NOT IN Y.

1 - RANDOM NODES
2 - RANGE OF NODES
3 - SPECIFIED PLANE
4 - ALL NODES
5 - EXIT

ENTER NODE SELECTION OPTION (1,2,...5) --------->4
```

```
ENTER CONSTRAINT DIRECTION CODE (THREE VALUES) ->0 0 0

1 - RANDOM NODES
2 - RANGE OF NODES
3 - SPECIFIED PLANE
4 - ALL NODES
5 - EXIT

ENTER NODE SELECTION OPTION (1,2,...5) --------->3

ENTER CONSTRAINT DIRECTION CODE (THREE VALUES) ->1 0 0

PLANE IS DEFINED BY  AX + BY + CZ = D.
  ENTER COEFFICIENTS ( A , B , C , D) ---------->1 0 0 0

            24 NODES FOUND

1 - RANDOM NODES
2 - RANGE OF NODES
3 - SPECIFIED PLANE
4 - ALL NODES
5 - EXIT

ENTER NODE SELECTION OPTION (1,2,...5) --------->3

ENTER CONSTRAINT DIRECTION CODE (THREE VALUES) ->0 1 0

PLANE IS DEFINED BY  AX + BY + CZ = D.
  ENTER COEFFICIENTS ( A , B , C , D) ---------->0 1 0 0

            14 NODES FOUND

1 - RANDOM NODES
2 - RANGE OF NODES
3 - SPECIFIED PLANE
4 - ALL NODES
5 - EXIT

ENTER NODE SELECTION OPTION (1,2,...5) --------->5

            39 BOUNDARY CONDITIONS ADDED
            39 BOUNDARY CONDITIONS TOTAL

:: LIST

LISTING OF DATA FILES AVAILABLE

#10 --- PLATE+BCS
#11 --- PLATE-4

:: DELE
ENTER LABEL                    ;;PLATE-4

:: PLOT
ENTER INPUT FILE #1 LABEL            ;;PLATE+BCS
 $VERT
  WHICH AXIS IS VERTICAL?
  ENTER 1 FOR X, 2 FOR Y, OR 3 FOR Z ..........: 1
 $EYE
  ENTER THE EYE POSITION,
  XEYE, YEYE, ZEYE ............................: 0 0 10000
 $DRAW

 AT END OF PLOT, ENTER CHARACTER TO CONTINUE.
 READY (Y,N) ? ....: Y
```



```
*ZOOM
  DO YOU WANT THE ZOOM FUNCTION (Y,N) ? ......: Y
  SCALE THE ZOOM AREA (Y,N) ? ................: Y
  DIGITIZE THE LOWER LEFT CORNER
  AND THE UPPER RIGHT CORNER OF THE
  ZOOM AREA
  READY?(Y,N) ................................: Y

*LABE
  LABEL THE ELEMENTS?(Y,N) ...................: Y
  LABEL THE NODES?(Y,N) ......................: Y
  LABEL ALL THE SURFACES?(Y,N) ...............: Y
*AXES
  PLOT AND LABEL THE AXES?(Y,N) ..............: N
*DRAW

 AT END OF PLOT, ENTER CHARACTER TO CONTINUE.
 READY (Y,N) ? ....: Y
```



```
$ZOOM
  DO YOU WANT THE ZOOM FUNCTION (Y,N) ? ......: N
$MAIN

:: LIST

LISTING OF DATA FILES AVAILABLE

$10 --- PLATE+BCS

:: LOAD
ENTER INPUT FILE $1 LABEL                    ;;PLATE+BCS
ENTER OUTPUT FILE LABEL                      ;;FINAL
CREATING NEW DATA FILE

LOAD SPECIFICATION OPTIONS
  N - NODAL LOAD ENTRY
  E - ELEMENT LOAD ENTRY
  L - LIST EXISTING LOADS
  H - PRINTS THIS LIST
  S - STOP LOAD ENTRY

ENTER LOAD SPECIFICATION OPTION    --->N

NODE SPECIFICATION OPTIONS
  S - SINGLE NODE
  R - RANGE OF NODES
  P - ALL NODES ON A GIVEN PLANE
  H - PRINTS THIS LIST
  E - EXIT NODAL LOAD SPECIFICATION SECTION

ENTER NODE SPECIFICATION OPTION    --->S
ENTER CASE NUMBER                  --->1
ENTER FORCE VECTOR (FX,FY,FZ)      --->0 100 0
NODE NUMBER TO BE LOADED ?         --->386

ENTER NODE SPECIFICATION OPTION    --->E

LEAVING NODAL LOAD SPECIFICATION SECTION

ENTER LOAD SPECIFICATION OPTION    --->H

LOAD SPECIFICATION OPTIONS
  N - NODAL LOAD ENTRY
  E - ELEMENT LOAD ENTRY
  L - LIST EXISTING LOADS
  H - PRINTS THIS LIST
  S - STOP LOAD ENTRY

ENTER LOAD SPECIFICATION OPTION    --->S

LEAVING LOAD SPECIFICATION MODULE

:: LIST

LISTING OF DATA FILES AVAILABLE

$10 --- PLATE+BCS
$11 --- FINAL

:: DELE
ENTER LABEL                 ;;PLATE+BCS

:: STOP

LISTING OF DATA FILES AVAILABLE

$11 --- FINAL


     STOP
     063700 MAXIMUM EXECUTION FL.
     29.561 CP SECONDS EXECUTION TIME.
FILE QUOTA EXCEEDED
COMMAND- FILES
--LOCAL FILES--
  TAPE14    XP       $INPUT    $OUTPUT    TAPE10
  TAPE12    TAPE13   TAPE11    TAPE16     TAPE15
  TAPE17
COMMAND- RETURN,TAPE10,TAPE12,TAPE13,TAPE14,TAPE15,TAPE16,TAPE17
```

7.1.4 REFMT Execution

```
COMMAND- REWIND,TAPE11
COMMAND- COPYBF,TAPE11,UNFMT
 EOI ENCOUNTERED AFTER COPY OF FILE
        0, RECORD  1
COMMAND- RETURN,TAPE11
COMMAND- BEGIN,REFMT,P.

****************************************

BEGIN REFMT   --   MAGNA INPUT GENERATOR

****************************************


** SUMMARY OF INITIAL SCAN OF DATA FILE **

------------------------------------------

NUMBER OF NODAL POINTS .......    382
NUMBER OF ELEMENTS (TOTAL) ...     85
NUMBER OF CONSTRAINT RECORDS..     39
NUMBER OF LINEAR CONSTRAINTS..      0
NUMBER OF NODAL LOADS ........      1
NUMBER OF ELEMENT LOADS ......      0
DISTINCT LOAD CASES / GROUPS..      1


    MAGNA         NUMBER OF     ELEMENTS WITH
ELEMENT TYPE      ELEMENTS      UNSPEC. MATRL

      9              85              85


ENTER A THREE-LINE PROBLEM TITLE   (UP TO 80 CHARACTERS PER LINE)

.........1.........2.........3.........4.........5.........6.........7.........8
  THIN PERFORATED PLATE - PLANE STRESS - LINEAR, STATIC ANALYSIS
  LENGTH = 20., WIDTH=10. FOR QUADRANT MODELED, HOLE RADIUS = 5.0
  9-NODE PLANE STRESS ELEMENTS, 85 ELEMENTS TOTAL

********************************************

 MAJOR SOLUTION OPTIONS AND PARAMETERS

********************************************

ENTER ANALYSIS TYPE      (LINEAR,NONLINEAR) -LINEAR

ENTER ANALYSIS SUBTYPE   (STATIC,DYNAMIC)-----STATIC

INCLUDE THERMAL EFFECTS (Y/N) --------------N

POSTPROCESSOR FILE TO BE WRITTEN (Y/N) -----Y

********************************************
     END OF OPTIONS SPECIFICATIONS
********************************************
```

```
INDIVIDUAL ELEMENTS IN THE MODEL CONTAINUNDEFINED PROPERTIES

ELEMENT TYPE      =     9
NUMBER OF ELEMENTS=    85

PLEASE DEFINE A DEFAULT PROPERTY CODE FOR THIS ELEMENT TYPE,
OR ENTER MATERIALS DATA DIRECTLY BELOW.


MATERIAL PROPERTY DEFINITION OPTIONS

 E -- ENTER PROPERTY DATA DIRECTLY
 C -- SPECIFY A LIBRARY PROPERTY CODE
 L -- LIST SELECTED LIBRARY ENTRIES

ENTER OPTION  ( E , C , L ) -----------L

LIBRARY MATERIAL DESCRIPTIONS CAN BE LISTED BY MATERIAL TYPE
VALID MATERIAL TYPES ARE AS FOLLOWS ----

ACRYL - ACRYLICS
ALUMI - ALUMINUM ALLOYS
CASTI - CAST IRONS
COPPR - COPPER-BASED ALLOYS
GLASS - GLASSES
MAGNS - MAGNESIUM ALLOYS
NICKL - NICKEL ALLOYS
PLYMR - POLYMERIC MATERIALS
POLYC - POLYCARBONATES
STEEL - CARBON STEELS
STSTL - STAINLESS STEELS
TITNM - TITANIUM

ENTER MATERIAL TYPE (STEEL,STSTL,ETC.)--ALUMI


MAT.. CODE   ...........DESCRIPTION................

    00200    ALUMI - UNS A9 2024-0
    00202    ALUMI - UNS A9 2024-T36
    00205    ALUMI - UNS A9 2024-T62  SHEET
    00210    ALUMI - UNS A9 2021-T81  BAR
    00211    ALUMI - UNS A9 2021-T81  SHEET
    00220    ALUMI - UNS A9 5052-0
    00223    ALUMI - UNS A9 5052-H32
    00230    ALUMI - UNS A9 6061-0
    00231    ALUMI - UNS A9 6061-T4
    00232    ALUMI - UNS A9 6061-T6
    00240    ALUMI - UNS A9 7075-T6

MATERIAL PROPERTY DEFINITION OPTIONS

 E -- ENTER PROPERTY DATA DIRECTLY
 C -- SPECIFY A LIBRARY PROPERTY CODE
 L -- LIST SELECTED LIBRARY ENTRIES

ENTER OPTION  ( E , C , L ) -----------C

ENTER LIBRARY PROPERTY CODE --205


MATERIAL PROPERTIES DEFINITION FOR THE MODEL IS COMPLETE.
AT THIS POINT MATERIALS DATA MAY BE EDITED AS NECESSARY.
(NOTE THAT SOME DATA WHICH IS UNIMPORTANT FOR THE CURRENT
ANALYSIS MAY BE DEFINED AS ZERO)

CURRENT PROPERTIES ARE LISTED BELOW ----


 CODE  MODULUS   POIS.RATIO  DENSITY   YIELD STR. THERM.EXP.

   -9  .1050E+08  .3300E+00  .2590E-03  .4900E+05  .1260E-04

L = (L)IST CURRENT PROPERTIES TABLE
C = (C)HANGE AN ENTRY IN THE TABLE
S = (S)TOP EDITING

ENTER OPTION  ( L , C , S ) -----------S
```

```
****************************************
*                                      *
*       DATA GENERATION COMPLETE       *
*                                      *
****************************************

     STOP
     065500 MAXIMUM EXECUTION FL.
      1.113 CP SECONDS EXECUTION TIME.
COMMAND- FILES
 --LOCAL FILES--
 *P          $INPUT      $OUTPUT     UNFMT      FDATA
  ISCR1
COMMAND- RETURN,UNFMT,ISCR1
COMMAND- REWIND,FDATA
COMMAND- REQUEST,DATA,*PF.
COMMAND- COPYBF,FDATA,DATA
COMMAND- CATALOG,DATA,SHEETWITHHOLE,CY=1,RP=900,ID=BROCKMAN
 INITIAL CATALOG
 CT ID= BROCKMAN PFN=SHEETWITHHOLE
 CT CY= 001 SN=AFFDL    0000002944 WORDS.
```

7.2 EXECUTION OF CORGEN / EXPAND / PREP / REFMT SEQUENCE

This Subsection contains an example of the use of CORGEN to generate a finite element mesh from coordinate data in lofting form, which in this instance is stored on a data file. The EXPAND utility is used to translate the surface model output by CORGEN to three-dimensional form. PREP is used to refine the mesh and specify constraints. REFMT generates the formatted data deck ready for analysis.

The geometric data used in preparing this model is lofting data describing the forward canopy of the T-38 aircraft. Near the forward edge, the canopy rests on a metallic arch structure which does not provide for any bolted attachment; the final model will contain a rather small row of elements along this boundary, which could be used to generate contact elements for detailed analysis of the true support condition.

7.2.1 Summary of Modelling Procedure

The modelling data provided for the T-38 airplane canopy consists of nine lofting planes with seven points defined on each plane. This data was entered to CORGEN as a file structured so that each lofting line corresponded to one group of data. The groups were input such that the first group represented the first lofting line at the fore point on the canopy and the ninth group represented the aft area of the canopy.

The CORGEN program was executed to convert the lofting data into the preprocessor internal format description. The program execution was uninvolved for this model as the default options for group formulation and element generation were utilized. The sequence of commands to create the model are identified below:

1 -	NODE; FILE; READ; EXIT	- node input from file,
2 -	ELEM; GENE; EXIT	- element generation,
3 -	FINI	- write model data to file.

These operations resulted in a 2-D surface model consisting of twelve nine node elements.

The EXPAND program was executed to translate the surface model to a three-dimensional solid element model. There exists only the option of selecting the direction to expand the surface in this program.

The PREP program was executed to perform several functions:

- verification of the lofting model data,
- mask to 16 node elements (from 27 node elements),
- refine the model,
- optimize the node numbering for the analysis,
- select 14 point integration rule for analysis, and
- specify boundary conditions.

The sequence of steps to execute PREP are outlined below and listed in the following paragraphs of this section.

STEP 1: Verification of CORGEN/EXPAND model
- PLOT (plot original canopy model)
- PLOT (find local element orientation axes)

STEP 2: Create 16-node solid elements from 27 node model
- MASK (eliminate extraneous nodes)

STEP 3: Refine the model
- REFINE (execute 2 times for R direction)
- PLOT (verify refinement, check new element numbering for next refinement)
- REFINE (only selected elements in S direction)

STEP 4: Optimize node numbering
- RENUMBER (repeated to insure optimal numbering)
- PLOT (verify model geometry, and identify node numbers for boundary conditions)

STEP 5: Select element properties
- PROPS (select 14 point integration rule, remainder of properties given in REFMT)

STEP 6: Specify boundary conditions
- PLOT (ZOOM to find nodes to be constrained)
- BOUNDS (constrain nodal degrees of freedom)

A natural frequencies analysis will be performed, so no loads data are specified at this time.

The REFMT program was executed to prepare a data file for MAGNA analysis. This section details data specification for a natural frequency analysis of the model. The REFMT program has interpreted the 16 node elements as MAGNA type 8 element of which there are 45 elements. The sample run illustrates the definition of undefined material property types. A final file is generated for a MAGNA analysis.

Sections 7.2.2 through 7.2.5 list the program executions of the CORGEN / EXPAND / PREP /REFMT sequence for completing the T-38 aircraft windshield model.

7.2.2 CORGEN Execution

```
COMMAND- ATTACH,P,PREPROCESSORPROC,ID=BROCKMAN,MR=1.
 AT CY= 993 SN=AFFDL
COMMAND- ATTACH,TAPE11,WINDSHIELDDATA,CY=1.
COMMAND- FILES
--LOCAL FILES--
 $INPUT    $OUTPUT   $P        $TAPE11
COMMAND- BEGIN,CORGEN,P.

   ***********************************
   ********** BEGIN CORGEN **********
   ***COARSE GRID INPUT PREPROCESSOR***
   ***********************************

  FOR A LISTING OF COMMANDS TYPE 'COMM'
    COMMAND..: COMM

    (HELP)
    (NODE) DATA
    (ELEM)ENT DATA
    (GROU)P DATA
    (SAVE) CURRENT MODEL
    (REST)ORE MODEL
    (COOR)DINATE SYSTEM
    (INTE)RPOLATE DATA
    (COMM)ANDS LISTING
    (DISP)LAY MODEL
    (FINI)SH MODEL
    (THIC)KNESS
    (STOP)
    COMMAND..: NODE
   OPTION?..: COMM
   NODE INPUT/EDIT MODULE
   COMMAND OPTIONS:
    (COMM)ANDS
    (HELP)
    (LIST)
    (TABL)E
    (GENE)RATE
    (DELE)TE
    (ALTE)R
    (INTE)RPOLATE
    (FILE) INPUT OF COORDS
    (KEYB)OARD INPUT OF COORDS
    (EXIT)
   OPTION?..: FILE
   ENTER TAPE NUMBER OF FILE
   CONTAINING DATA (11-19).: 11
   OPTION?.: COMM
   NODE ENTRY OPTIONS..:
    (COMM)ANDS LISTING
    (HELP)
    (READ) VALUES
    (COOR)DINATE SYSTEM
    (TRAN)SLATE COORDS
    (DEFA)ULT OPTIONS
    (FILE) DEFINITION
    (REWI)ND FILE
    (SKIP) RECORDS ON FILE
    (EXIT)
   OPTION?.: READ

   THE USER MUST SPECIFY
   TWO PARAMETERS TO CONTROL
   NODE INPUT:
     $ OF NODES/GROUP (NNG)
     $ OF GROUPS (NG).
   THERE MUST BE AT LEAST
   NNG X NG NODES PRESENT.
   ENTER $ NODES PER GROUP.: 7
   ENTER $ GROUPS TO BUILD.: 9
```

```
GROUP   BEG. NODE   END NODE
  1         1          7
  2         8         14
  3        15         21
  4        22         28
  5        29         35
  6        36         42
  7        43         49
  8        50         56
  9        57         63
TOTAL NODES INPUT =    63
TOTAL GROUPS INPUT =    9
  OPTION?.: EXIT
  OPTION?..: COMM
  NODE INPUT/EDIT MODULE
  COMMAND OPTIONS:
   (COMM)ANDS
   (HELP)
   (LIST)
   (TABL)E
   (GENE)RATE
   (DELE)TE
   (ALTE)R
   (INTE)RPOLATE
   (FILE) INPUT OF COORDS
   (KEYB)OARD INPUT OF COORDS
   (EXIT)
 OPTION?..: EXIT
  COMMAND..: COMM

   (HELP)
   (NODE) DATA
   (ELEM)ENT DATA
   (GROU)P DATA
   (SAVE) CURRENT MODEL
   (REST)ORE MODEL
   (COOR)DINATE SYSTEM
   (INTE)RPOLATE DATA
   (COMM)ANDS LISTING
   (DISP)LAY MODEL
   (FINI)SH MODEL
   (THIC)KNESS
   (STOP)
   COMMAND..: ELEM

 OPTION?.: COMM

 ELEMENT EDIT/GENERATOR
 COMMANDS AVAILABLE:
  (LIST)
  (TABL)E OF ELEMENTS
  (GENE)RATE
  (DELE)TE
  (ALTE)R
  (INTE)RPOLATE
  (FILE) INPUT OF CONNECTIVITY
  (KEYB)OARD INPUT OF CONNECTIVITY
  (HELP)
  (COMM)ANDS
  (EXIT)

 OPTION?.: GENE
 INTERPOLATE NODES PRIOR TO
 ELEMENT GENERATION (Y/N) : N

 OPTION?.: EXIT
  COMMAND..: COMM
```

```
    (HELP)
    (NODE) DATA
    (ELEM)ENT DATA
    (GROU)P DATA
    (SAVE) CURRENT MODEL
    (REST)ORE MODEL
    (COOR)DINATE SYSTEM
    (INTE)RPOLATE DATA
    (COMM)ANDS LISTING
    (DISP)LAY MODEL
    (FINI)SH MODEL
    (THIC)KNESS
    (STOP)
    COMMAND..: FINI
     STOP
     065600 MAXIMUM EXECUTION FL.
      0.361 CP SECONDS EXECUTION TIME.
COMMAND- FILES
--LOCAL FILES--
 $INPUT    $OUTPUT   *P        *TAPE11    TAPE9
  TAPE10
COMMAND- RETURN,TAPE11,TAPE9
```

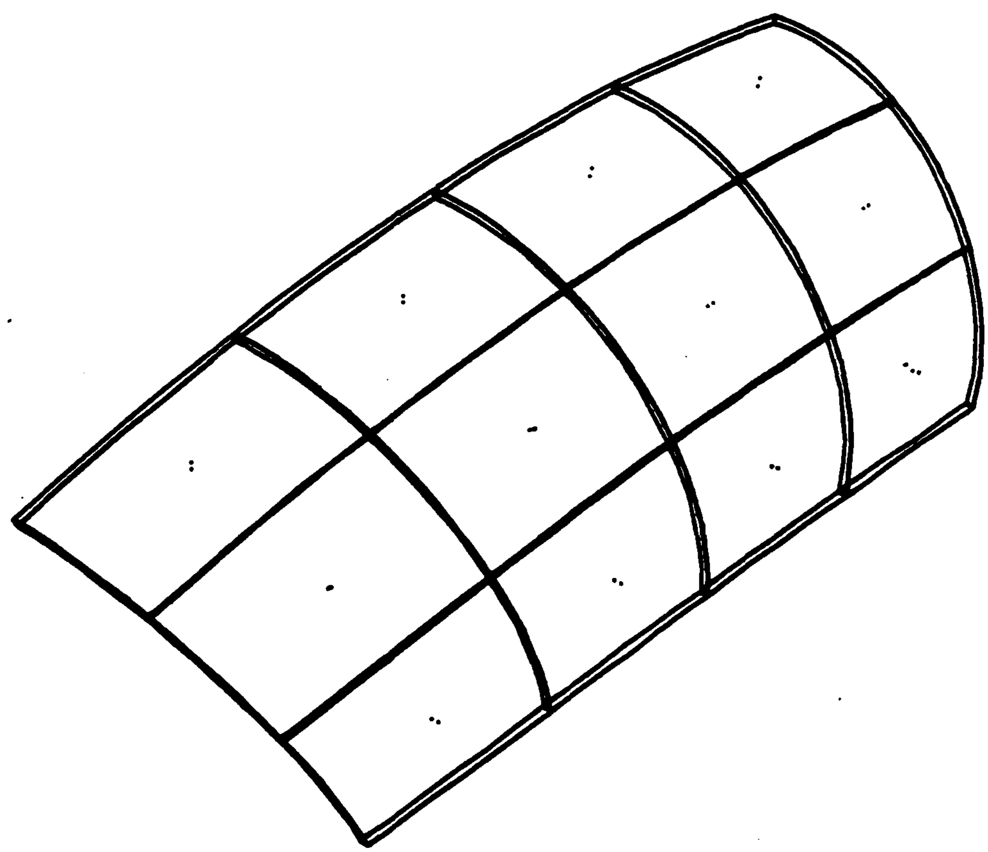
7.2.3 EXPAND Execution

```
COMMAND- FILES
--LOCAL FILES--
 $INPUT     $OUTPUT   ZZ          TAPE10
COMMAND- BEGIN,EXPAND,P.

*************************************
*                                   *
*           BEGIN EXPAND            *
* THICKNESS EXPANSION PREPROCESSOR *
*                                   *
*************************************

 THICKNESS EXPANSION OPTIONS:
   1 - EXPAND FROM BOTTOM - UP
   2 - EXPAND ABOUT MIDPLANE
   3 - EXPAND FROM TOP - DOWN
 ENTER OPTION (1,2,3)........: 2


 DO YOU WISH A LISTING OF
 EXPANSION DATA (Y,N)?...: N
     STOP
     034700 MAXIMUM EXECUTION FL.
      0.357 CP SECONDS EXECUTION TIME.
COMMAND- FILES
--LOCAL FILES--
  TAPE12   $INPUT    $OUTPUT   ZZ          TAPE10
  TAPE11
COMMAND- RETURN,TAPE10,TAPE12
```

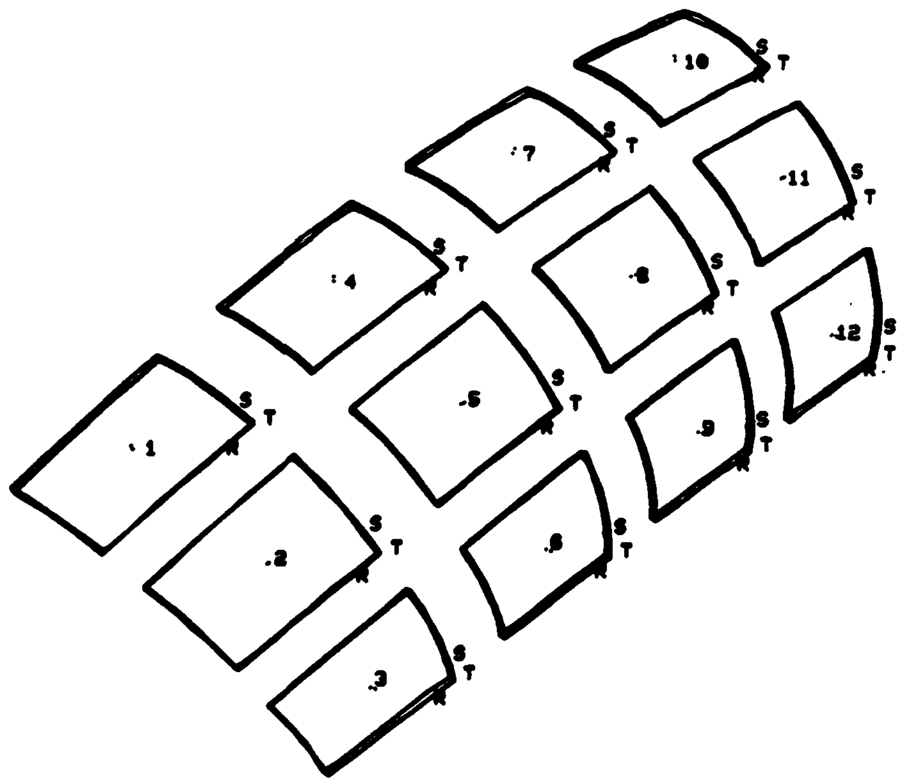
7.2.4 PREP Execution

```
COMMAND- FILES
--LOCAL FILES--
 $INPUT    $OUTPUT   XP        TAPE11
COMMAND- BEGIN,PREP,P

          ****************************************
          *************  BEGIN PREP  *************
          **** MAGNA PREPROCESSING UTILITIES *****
          ****************************************

PLOTTING TERMINAL ( Y , N ) ?;;Y
TEKTRONIX OR HP ( T , H ) ?  ;;T

TEKTRONIX TERMINAL TYPES ----
 0, 4006-1
 1, 4010 / 4012 / 4013 / 4052
 2, 4014 / 4015
 3, 4014 / 4015   ( ENH. GR. MOD. )
 4, 4114

ENTER TERMINAL TYPE ( 0 - 4 );;3
ENTER CHARACTERS PER SECOND  ;;120
NUMBER OF INPUT DATA FILES ? ;;1
ENTER FILE #                 ;;11
ENTER LABEL                  ;;T30EGRD

TYPE 'HELP' FOR LIST OF COMMANDS

:: HELP

AVAILABLE COMMANDS :
BOUNDS    - SETS BOUNDARY CONDITION DATA
CONTAC    - CREATES CONTACT ELEMENTS FROM FACES OF SOLIDS
COPY      - COPIES DATA TO SECOND FILE
CREATE    - CREATES NEW MODEL 'FROM SCRATCH'
DELETE    - RELEASES MODEL FILES FOR GENERAL USE
EDIT      - ALLOWS DIRECT EDITING OF DATA FILES
FILL      * ADDS NEW NODES TO EXISTING ELEMENTS
HELP      - PRINTS THIS LIST
LINEAR    - DEFINE LINEAR (MULTIPOINT) CONSTRAINTS
LIST      - LISTING OF AVAILABLE DATA FILES
LOAD      - ADDS PHYSICAL LOADING DATA
MASK      - DELETES NODES FROM EXISTING ELEMENTS
MERGE     * MERGES TWO DATA FILES INTO ONE MODEL
NAME      - USER ROUTINE TO NAME MODEL FILES
PLOT      - PLOTS SELECTED MODEL IN THREE SPACE
PRINT     - PRINTS DATA AT TERMINAL
PROPS     - ADDS PHYSICAL PROPERTIES DATA
REFINE    * REFINES SELECTED ELEMENTS
REFLECT   - REFLECTS MODEL ACROSS USER DEFINED PLANE
RENUMBER  * OPTIMIZES NODE POINT NUMBERING
ROTATE    - ROTATES MODEL ABOUT COORDINATE AXES
SHELL     - DEFINES ELEMENTS AS SHELL OR 3-D
SIFT      * ELIMINATES UNUSED NODE NUMBERS
STOP      - ENDS PROGRAM OPERATION
TRANSLATE - TRANSLATES MODEL POSITION IN 3-SPACE
TIDY      * ELIMINATES DUPLICATED NODE NUMBERS
TIME      - PRINTS CP TIME SINCE SESSION BEGAN
TOLER     - CHANGES DEFAULT TOLERANCE FACTOR

 * -> DOES NOT RETAIN BOUNDARY CONDITION, LOADING, OR
      LINEAR CONSTRAINT DATA.

SPECIFIC INFORMATION ON COMMANDS CAN BE OBTAINED BY
  BY ENTERING '?' WHILE IN COMMAND MODE.

:: PLOT
ENTER INPUT FILE #1 LABEL               ;;T30EGRD

INITIALIZATION OF PLOTTING ROUTINE ERASES SCREEN.
READY (Y,N) ? ....: Y
```

```
*DRAW

 AT END OF PLOT, ENTER CHARACTER TO CONTINUE.
 READY (Y,N) ? ....: Y
```

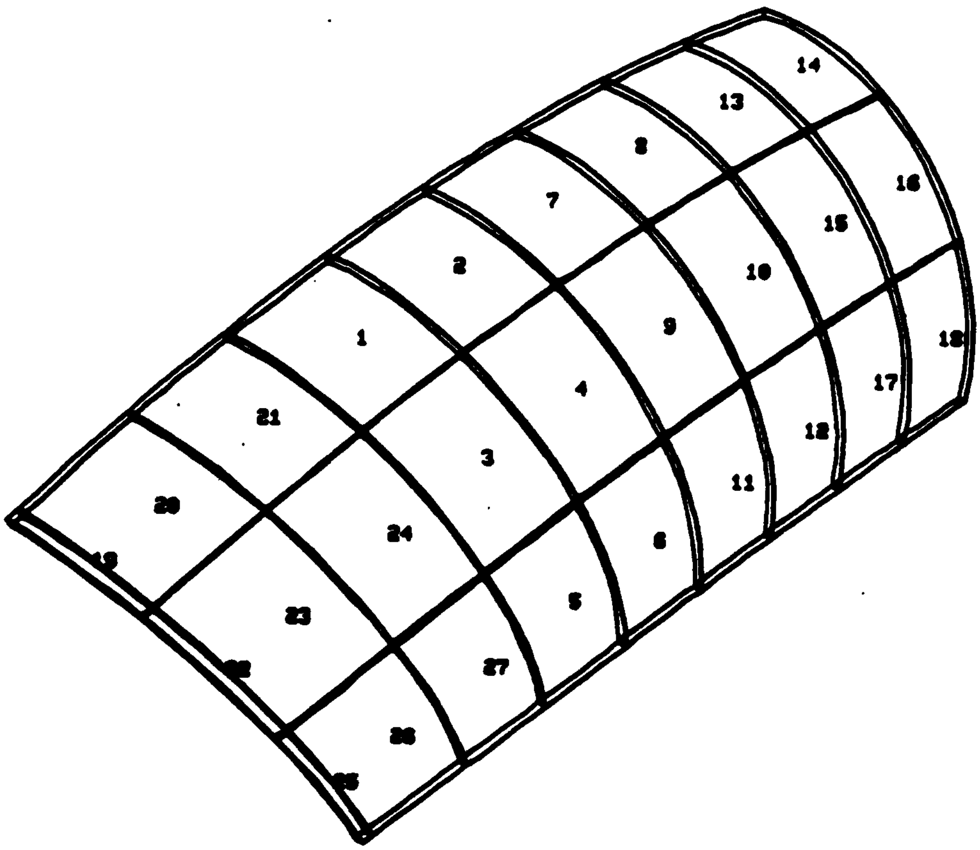


```
*SHRINK
  ENTER THE SHRINK SCALE(0.-1.) ...............: 0.3
*ORIENT
  PLOT ORIENTATION AXES?(Y,N) .................: Y
*LABEL
  LABEL THE ELEMENTS?(Y,N) ....................: Y
  LABEL THE NODES?(Y,N) .......................: N
*DRAW

 AT END OF PLOT, ENTER CHARACTER TO CONTINUE.
 READY (Y,N) ? ....: Y
```



```
?MAIN

:: MASK
ENTER INPUT FILE #1 LABEL                 ;;T38EGRD
ENTER OUTPUT FILE LABEL                   ;;16NODES
CREATING NEW DATA FILE
MASK OPTIONS -
  1. 3-D ELEMENTS ONLY
  2. 2-D ELEMENTS ONLY
ENTER OPTION...;;1
MAX NODE POSITION # IN FINAL ELEMENTS?
  (8,16,20,26,27) ;;16

MASK COMPLETE.
PERFORM SUBSEQUENT SIFT ? (Y,N) ->Y

     SIFT COMPLETE
  24 UNUSED NODES DELETED

:: REFINE
ENTER INPUT FILE #1 LABEL                 ;;16NODES
ENTER OUTPUT FILE LABEL                   ;;REFIN1
CREATING NEW DATA FILE

  BEGIN REFINE ROUTINE...

1 - RANDOM ELEMENT INPUT
2 - RANGE OF ELEMENTS
3 - EXECUTE REFINEMENT
ENTER OPTION --------------->2
MAXIMUM NUMBER OF ELEMENTS
  SPECIFIABLE -------------->  150
NUMBER STILL AVAILABLE ----->  150

ENTER BEGINNING ELEMENT NUMBER, ENDING ELEMENT
  NUMBER, AND INCREMENT FOR RANGE.
ENTER NBEG,NEND,INCR ------->4,12,1

1 - RANDOM ELEMENT INPUT
2 - RANGE OF ELEMENTS
3 - EXECUTE REFINEMENT
ENTER OPTION --------------->3

ENTER CUT DIRECTION (R,S,T) >R

MAXIMUM CUTS PER ELEMENT ---> 5
ENTER CUTS PER ELEMENT ----->1

EQUALLY SPACED CUTS ? (Y,N) >Y

REFINE COMPLETE.
PERFORM SUBSEQUENT TIDY ? (Y,N) ->Y

TIDY OPTION ONE OR TWO ? (OPT1, OPT2, HELP) ->HELP

TIDY OPTION ONE WILL WORK ON ALL MODELS GIVEN
  ENOUGH TIME.
TIDY OPTION TWO INCLUDES POSITIONALLY DEPENDENT
  REORDERING OF NODES AND WORKS FASTER ON MOST
  MODELS.
SINCE OPTION TWO CAN CONCEIVABLY ATTEMPT RENUM-
  BERING FROM ALL THREE COORDINATE DIRECTIONS AND
  EVENTUALLY CALL OPTION ONE IF NECESSARY, OPTION
  ONE MAY BE FASTER THAN OPTION TWO IN SOME CASES
IT IS SUGGESTED THAT OPTION TWO BE TRIED FIRST
  AND OPTION ONE BE USED IF THE MESSAGE

   'OPTION TWO FAILED.  OPTION ONE BEING INVOKED'

  IS DISPLAYED.

TIDY OPTION ONE OR TWO ? (OPT1, OPT2, HELP) ->OPT2

     TIDY COMPLETE
  60 DUPLICATE NODES DELETED
```

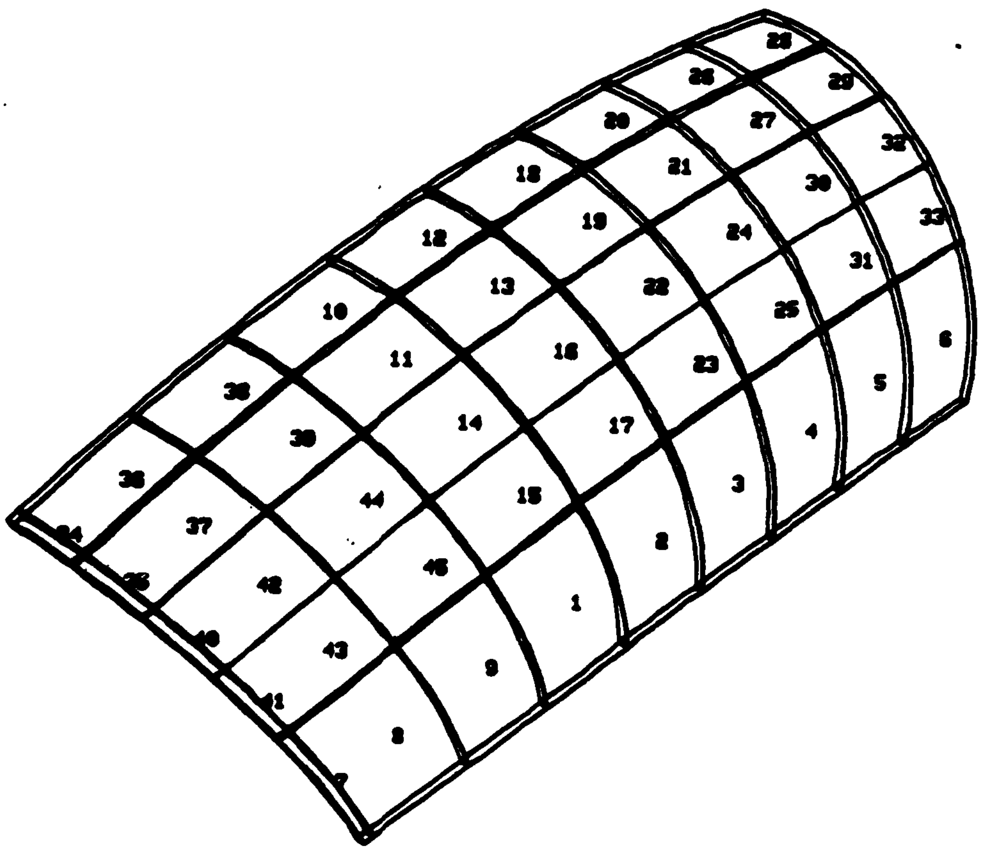
```
:: REFINE
ENTER INPUT FILE #1 LABEL                    ;;REFIN1
ENTER OUTPUT FILE LABEL                      ;;REFIN2
CREATING NEW DATA FILE

  BEGIN REFINE ROUTINE...

1 - RANDOM ELEMENT INPUT
2 - RANGE OF ELEMENTS
3 - EXECUTE REFINEMENT
ENTER OPTION -------------->2
MAXIMUM NUMBER OF ELEMENTS
  SPECIFIABLE -------------->  150
NUMBER STILL AVAILABLE ----->  150

ENTER BEGINNING ELEMENT NUMBER, ENDING ELEMENT
  NUMBER, AND INCREMENT FOR RANGE.
ENTER NBEG,NEND,INCR ------->1,3,1

1 - RANDOM ELEMENT INPUT
2 - RANGE OF ELEMENTS
3 - EXECUTE REFINEMENT
ENTER OPTION -------------->3

ENTER CUT DIRECTION (R,S,T) >R

MAXIMUM CUTS PER ELEMENT ---> 5
ENTER CUTS PER ELEMENT ----->2

EQUALLY SPACED CUTS ? (Y,N) >N

ENTER CUT POSITIONING IN
  ASCENDING ORDER (0-100) -->45,95

REFINE COMPLETE.
PERFORM SUBSEQUENT TIDY ? (Y,N) ->Y

TIDY OPTION ONE OR TWO ? (OPT1, OPT2, HELP) ->OPT2

      TIDY COMPLETE
   20 DUPLICATE NODES DELETED

:: LIST

LISTING OF DATA FILES AVAILABLE

#10 --- 16NODES
#11 --- T30EGRD
#12 --- REFIN1
#13 --- REFIN2

:: DELETE
ENTER LABEL                    ;;16NODES

:: DELETE
ENTER LABEL                    ;;T30EGRD

:: DELETE
ENTER LABEL                    ;;REFIN1

:: PLOT
ENTER INPUT FILE #1 LABEL                    ;;REFIN2
 %ORIENT
   PLOT ORIENTATION AXES?(Y,N) ..................: N
 %SHRINK
   ENTER THE SHRINK SCALE(0.-1.) ...............: 0
 %DRAW

  AT END OF PLOT, ENTER CHARACTER TO CONTINUE.
  READY (Y,N) ? ....: Y
```



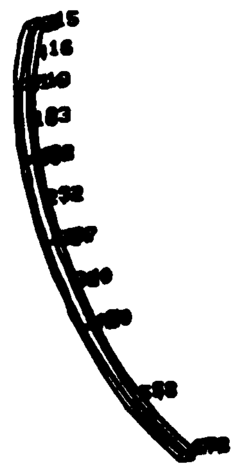
14
13
8
7
2
1
16
15
10
9
4
21
3
18
17
12
11
20
24
6
5
23
27
26

```
:MAIN

:: REFINE
ENTER INPUT FILE #1 LABEL                    ;;REFIN2
ENTER OUTPUT FILE LABEL                      ;;REFIN3
CREATING NEW DATA FILE

  BEGIN REFINE ROUTINE...

1 - RANDOM ELEMENT INPUT
2 - RANGE OF ELEMENTS
3 - EXECUTE REFINEMENT
ENTER OPTION --------------->1
MAXIMUM NUMBER OF ELEMENTS
  SPECIFIABLE ------------->  150
NUMBER STILL AVAILABLE ----->  150

ENTER ELEMENT NUMBER AND <RETURN> FOR EACH
  ELEMENT TO BE REFINED.  ELEMENTS NEED NOT BE
  ENTERED IN ASCENDING ORDER.  ELEMENT NUMBER 0
  TERMINATES INPUT.
ENTER ELEMENT # ----------->19
ENTER ELEMENT # ----------->20
ENTER ELEMENT # ----------->21
ENTER ELEMENT # ----------->1
ENTER ELEMENT # ----------->2
ENTER ELEMENT # ----------->7
ENTER ELEMENT # ----------->8
13
ENTER ELEMENT # ----------->13
ENTER ELEMENT # ----------->14
ENTER ELEMENT # ----------->22
ENTER ELEMENT # ----------->23
ENTER ELEMENT # ----------->24
ENTER ELEMENT # ----------->3
ENTER ELEMENT # ----------->4
ENTER ELEMENT # ----------->9
ENTER ELEMENT # ----------->10
ENTER ELEMENT # ----------->15
ENTER ELEMENT # ----------->16
ENTER ELEMENT # ----------->0

1 - RANDOM ELEMENT INPUT
2 - RANGE OF ELEMENTS
3 - EXECUTE REFINEMENT
ENTER OPTION --------------->3

ENTER CUT DIRECTION (R,S,T) >S

MAXIMUM CUTS PER ELEMENT ---> 5
ENTER CUTS PER ELEMENT ----->1

EQUALLY SPACED CUTS ? (Y,N) >Y

REFINE COMPLETE.
PERFORM SUBSEQUENT TIDY ? (Y,N) ->Y

TIDY OPTION ONE OR TWO ? (OPT1, OPT2, HELP) ->OPT2

     TIDY COMPLETE
  136 DUPLICATE NODES DELETED

:: RENUMBER
ENTER INPUT FILE #1 LABEL                    ;;REFIN3
ENTER OUTPUT FILE LABEL                      ;;RENUM4
CREATING NEW DATA FILE

 NODE POINT REORDERING COMPLETE

 NODAL BANDWIDTH (OLD) =   70
 NODAL BANDWIDTH (NEW) =   70

:: TIME
   THE CPU TIME FROM THE START OF THIS SESSION IS    12.256 SEC.
```

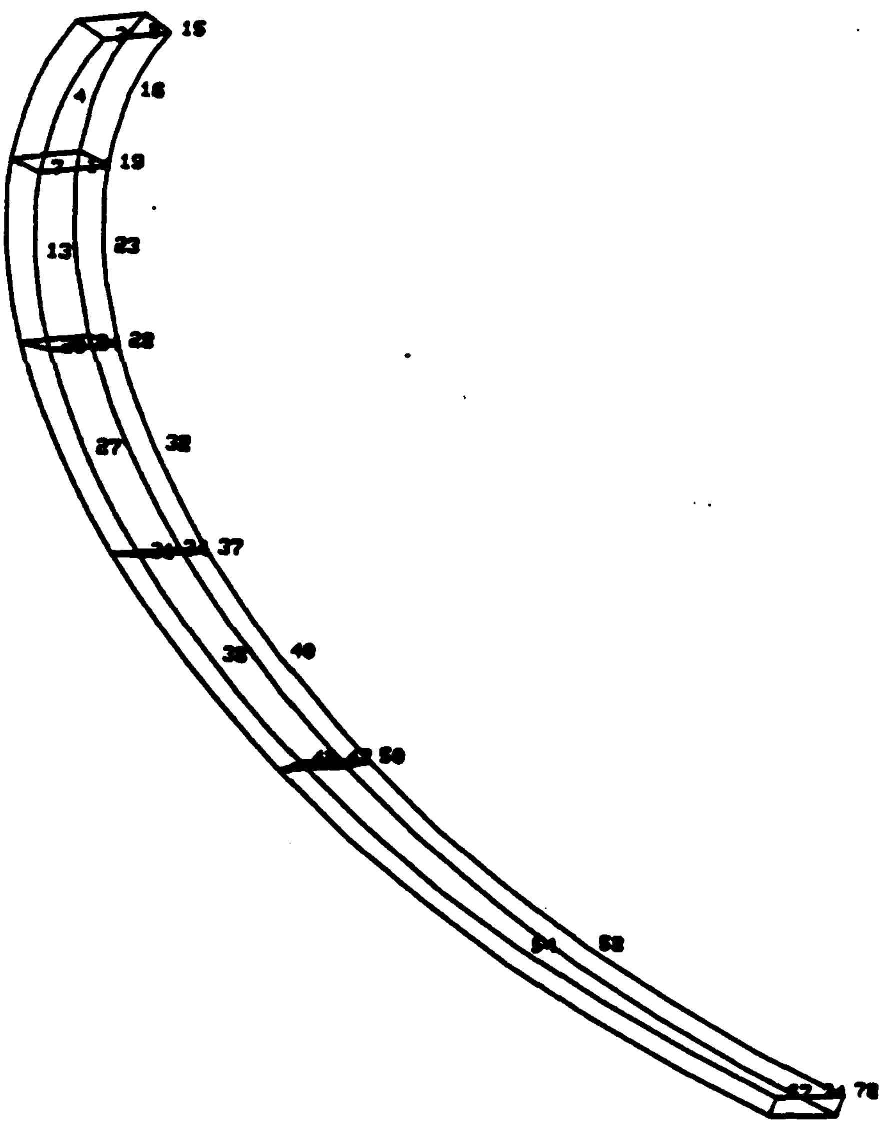
```
:: LIST

LISTING OF DATA FILES AVAILABLE

$10 --- REFIN3
$11 --- RENUMB4
$13 --- REFIN2

:: DELETE
ENTER LABEL                    ;;REFIN2

:: DELETE
ENTER LABEL                    ;;REFIN3

:: RENUMBER
ENTER INPUT FILE $1 LABEL               ;;RENUMB4
ENTER OUTPUT FILE LABEL                 ;;RENUMB5
CREATING NEW DATA FILE

 NODE POINT REORDERING COMPLETE

 NODAL BANDWIDTH (OLD) =   70
 NODAL BANDWIDTH (NEW) =   70

:: DELETE
ENTER LABEL                    ;;RENUMB5

:: PLOT
ENTER INPUT FILE $1 LABEL               ;;RENUMB4
 IDRAW

  AT END OF PLOT, ENTER CHARACTER TO CONTINUE.
  READY (Y,N) ? ....: Y
```



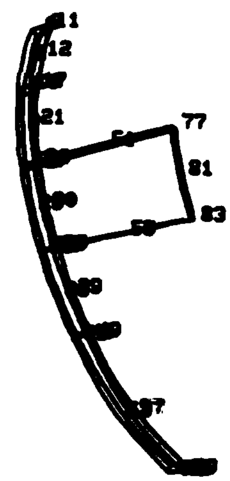
```
$LABEL
  LABEL THE ELEMENTS?(Y,N) ....................: N
  LABEL THE NODES?(Y,N) .......................: Y
  LABEL ALL THE SURFACES?(Y,N) ................: N
  ENTER THE SURFACE NO. (0-6, 0 PLOTS ALL) ....: 6
$HELP
  COMMAND         DESCRIPTION
  $$$$$$$         $$$$$$$$$$$
   AXES            AXES DRAW AND LABEL
   BOUNDS          PLOT BOUNDARY CONDITIONS
   CLIP            CLIP PLANE POSITION
   CUBE            SET MINIMA AND MAXIMA
   DEFAULT         SET DEFAULT VALUES
   DRAW            DRAW MODEL
   ELEMENTS        PLOTS ALL OR SELECTED ELEMENTS
   EYE             EYE POSITION
   HELP            LIST ALL COMMANDS
   LABELS          LABELS ELEMENTS AND/OR NODES
   MAIN            RETURNS CONTROL TO MAIN PROGRAM
   ORIENT          PLOTS R,S,T ORIENTATION AXES
   PROJECTION      PROJECTION TYPE
   REFLECT         REFLECT A PLANE
   ROTATE          ROTATE MODEL ABOUT AXES
   SCALE           SCALE PLOT
   SHRINK          SHRINK ELEMENTS
   SUMMARY         LIST ALL PARAMETER VALUES
   TIME            PRINT CPU TIME SINCE START OF SESSION
   TRANSLATE       TRANSLATE MODEL FROM ORIGIN
   VERTICAL        VERTICAL AXIS
   ZOOM            ZOOM ON THE MODEL
   SITE            SITE POSITION
$EYE
  ENTER THE EYE POSITION,
  XEYE, YEYE, ZEYE ............................: 100 200 100
$ELEM
  PLOT ALL THE ELEMENTS?(Y,N) .................: N
  ELEMENT INPUT OPTIONS
  OPTION 1:   RANDOM ELEMENTS(4,13,64,...)
              WHERE FIRST DIGIT = NO. ELEMENTS
  OPTION 2:   RANGE OF ELEMENTS(1,9,2 FOR 1-9,INC OF 2)
  ENTER THE INPUT OPTION(1,2) .................: 1
  ENTER THE ELEMENTS ..........................: 5 7,34,35,40,41
$DRAW

 AT END OF PLOT, ENTER CHARACTER TO CONTINUE.
 READY (Y,N) ? ....: Y
```



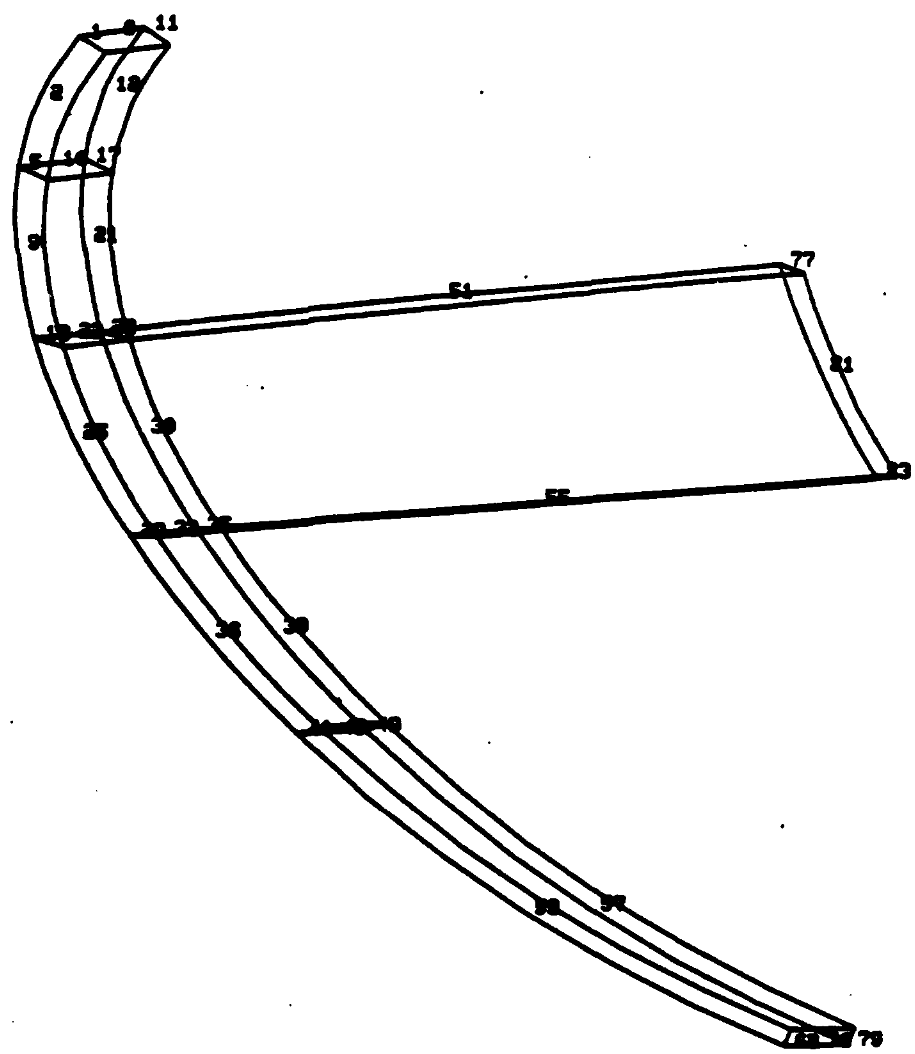
```
*ZOOM
  DO YOU WANT THE ZOOM FUNCTION (Y,N) ? ......: Y
  SCALE THE ZOOM AREA (Y,N) ? ................: N
  DIGITIZE THE LOWER LEFT CORNER
  AND THE UPPER RIGHT CORNER OF THE
  ZOOM AREA
  READY?(Y,N) ................................: Y

*DRAW

 AT END OF PLOT, ENTER CHARACTER TO CONTINUE.
 READY (Y,N) ? ....: Y
```



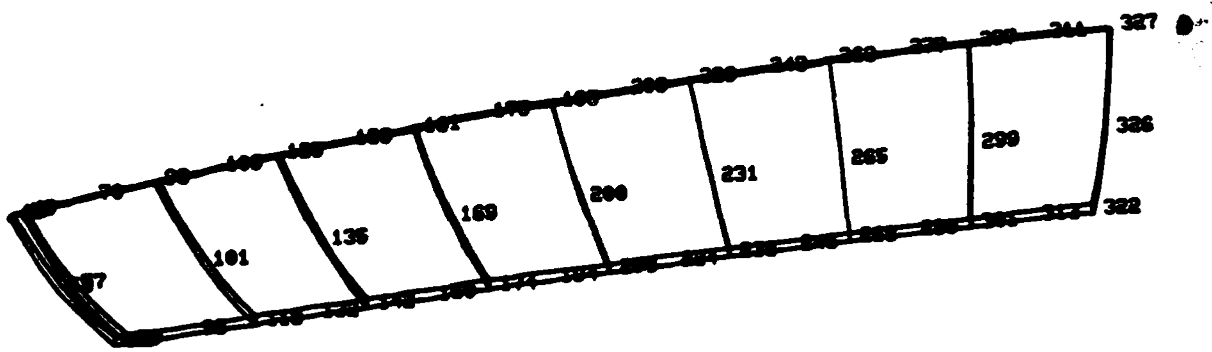
```
*ZOOM
  DO YOU WANT THE ZOOM FUNCTION (Y,N) ? ......: N
*ELEM
  PLOT ALL THE ELEMENTS?(Y,N) ..................: N
  ELEMENT INPUT OPTIONS
  OPTION 1:    RANDOM ELEMENTS(4,13,64,....)
               WHERE FIRST DIGIT = NO. ELEMENTS
  OPTION 2:    RANGE OF ELEMENTS(1,9,2 FOR 1-9,INC OF 2)
  ENTER THE INPUT OPTION(1,2) ..................: 1
  ENTER THE ELEMENTS ...........................: 6 7,34,35,40,41,42
*LABE
  LABEL THE ELEMENTS?(Y,N) .....................: N
  LABEL THE NODES?(Y,N) ........................: Y
  LABEL ALL THE SURFACES?(Y,N) .................: N
  ENTER THE SURFACE NO. (0-6, 0 PLOTS ALL) ....: 3
*DRAW

 AT END OF PLOT, ENTER CHARACTER TO CONTINUE.
 READY (Y,N) ? ....: Y
```



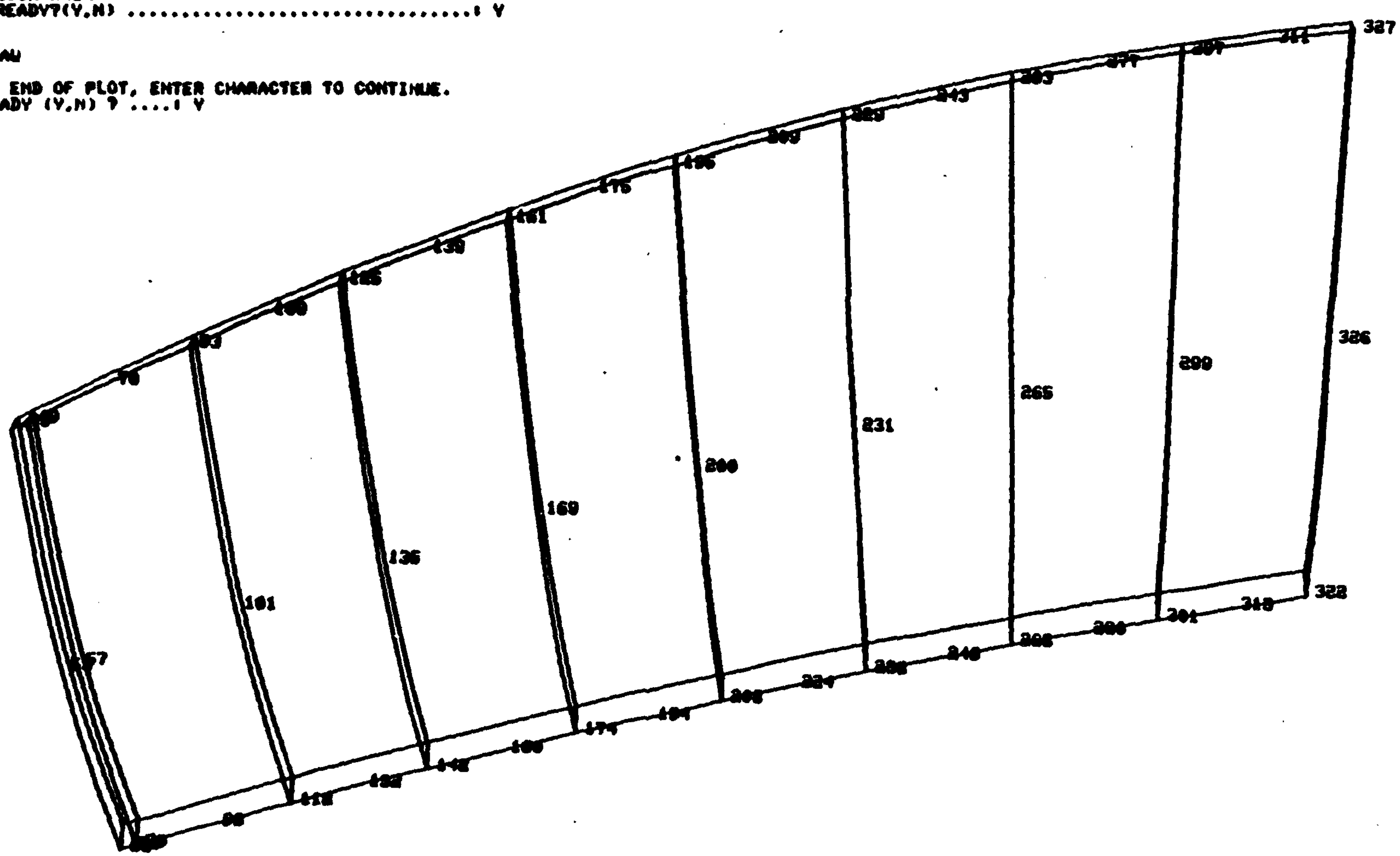
```
*ZOOM
  DO YOU WANT THE ZOOM FUNCTION (Y,N) ? ......: Y
  SCALE THE ZOOM AREA (Y,N) ? ................: N
  DIGITIZE THE LOWER LEFT CORNER
  AND THE UPPER RIGHT CORNER OF THE
  ZOOM AREA
  READY?(Y,N) ................................: Y

*DRAW

 AT END OF PLOT, ENTER CHARACTER TO CONTINUE.
 READY (Y,N) ? ....: Y
```



```
*FLEM
  PLOT ALL THE ELEMENTS?(Y,N) ..................: N
  ELEMENT INPUT OPTIONS
  OPTION 1:    RANDOM ELEMENTS(4,13,64,...)
               WHERE FIRST DIGIT = NO. ELEMENTS
  OPTION 2:    RANGE OF ELEMENTS(1,9,2 FOR 1-9,INC OF 2)
  ENTER THE INPUT OPTION(1,2) .................: 2
  ENTER THE ELEMENTS ..........................: 1,9,1
*ZOOM
  DO YOU WANT THE ZOOM FUNCTION (Y,N) ? ......: N
*DRAW

 AT END OF PLOT, ENTER CHARACTER TO CONTINUE.
 READY (Y,N) ? ....: Y
```



```
#ZOOM
  DO YOU WANT THE ZOOM FUNCTION (Y,N) ? ......: Y
  SCALE THE ZOOM AREA (Y,N) ? ................: N
  DIGITIZE THE LOWER LEFT CORNER
  AND THE UPPER RIGHT CORNER OF THE
  ZOOM AREA
  READY?(Y,N) ................................: Y

#DRAW

 AT END OF PLOT, ENTER CHARACTER TO CONTINUE.
 READY (Y,N) ? ....: Y
```



```
:ZOOM
  DO YOU WANT THE ZOOM FUNCTION (Y,N) ? ......: N
:MAIN

:: TIME
   THE CPU TIME FROM THE START OF THIS SESSION IS     16.058 SEC.

:: PROPS
ENTER INPUT FILE #1 LABEL                        ;;RENUMB4
ENTER OUTPUT FILE LABEL                          ;;PROPS
CREATING NEW DATA FILE

ELEMENT INPUT OPTIONS
  1)  ALL 2-D ELEMENTS
  2)  ALL 3-D ELEMENTS
  3)  ALL ELEMENTS
  4)  RANDOM ELEMENTS
  5)  RANGE OF ELEMENTS
ENTER OPTION...: 2

ELEMENT TYPES AVAILABLE...
   0 - DEFAULT (BAR,MEMBRANE,SOLID)
   1 - SHELL OR BEAM
   2 - PLANE STRESS (MEMBRANE)
   3 - PLANE STRAIN
   4 - AXISYMMETRIC
   5 - SHEAR PANEL
   6 - CONTACT ELEMENT
PRESENT ELEMENT TYPE =     0
CHANGE (Y,N) ..........: N

MATERIAL TYPE =     0
CHANGE (Y,N) ..........: N

INTEGRATION OPTIONS AVAILABLE -
   0 - DEFAULT FOR ELEMENT TYPE
   1 - SINGLE POINT
   6 - MIDFACE POINTS
   8 - 2 - PT. GAUSS
  14 - IRON'S RULE
  27 - 3 - PT. GAUSS
PRESENT INTEGRATION ORDER =     0
CHANGE (Y,N) ..........: Y
ENTER NEW INTEGRATION ORDER...: 14

ELEMENT PRINTING OPTIONS ----
   0 - PRINTING ON
   1 - PRINTING OFF

PRESENT PRINTING OPTION =  0
CHANGE ( Y, N ) ......: N



:: BOUNDS
ENTER INPUT FILE #1 LABEL                        ;;PROPS
ENTER OUTPUT FILE LABEL                          ;;BOUNDS
CREATING NEW DATA FILE

BOUNDARY CONDITIONS  MAY  BE ENTERED  FOR  RANDOM
  NODES,  RANGES OF NODES,  ALL NODES ON A PLANE,
  OR THE ENTIRE MODEL.

FOR EACH GROUP OF NODES,  CONSTRAINTS  MAY BE SET
  IN ANY COMBINATION OF THE X, Y, & Z DIRECTIONS.
  (NOTE THAT 2-D MODELS MUST BE FIXED IN Z)

CONSTRAINT DIRECTION CODE CONSTRUCTED AS FOLLOWS:

  ENTRY 1 - '1' FOR CONSTRAINED IN X DIRECTION
            '0' FOR UNCONSTRAINED IN X DIRECTION
  ENTRY 2 -     SAME FOR Y
  ENTRY 3 -     SAME FOR Z

  EXAMPLE -> '1,0,1' FOR CONSTRAINTS IN X AND Z
                     DIRECTIONS BUT NOT IN Y.
```

```
1 - RANDOM NODES
2 - RANGE OF NODES
3 - SPECIFIED PLANE
4 - ALL NODES
5 - EXIT

ENTER NODE SELECTION OPTION (1,2,...5) ---------->1

ENTER CONSTRAINT DIRECTION CODE (THREE VALUES) ->1 1 1

FOLLOW EACH NODE ENTERED BY TYPING 'RETURN'
ENTER '0' 'RETURN' TO END ENTRY.
ENTER NODE -->3
ENTER NODE -->4
ENTER NODE -->7
ENTER NODE -->13
ENTER NODE -->20
ENTER NODE -->27
ENTER NODE -->31
ENTER NODE -->38
ENTER NODE -->42
ENTER NODE -->54
ENTER NODE -->67
ENTER NODE -->1
ENTER NODE -->2
ENTER NODE -->5
ENTER NODE -->9
ENTER NODE -->18
ENTER NODE -->25
ENTER NODE -->29
ENTER NODE -->36
ENTER NODE -->44
ENTER NODE -->59
ENTER NODE -->68
ENTER NODE -->0

1 - RANDOM NODES
2 - RANGE OF NODES
3 - SPECIFIED PLANE
4 - ALL NODES
5 - EXIT

ENTER NODE SELECTION OPTION (1,2,...5) ---------->1

ENTER CONSTRAINT DIRECTION CODE (THREE VALUES) ->0 1 0

FOLLOW EACH NODE ENTERED BY TYPING 'RETURN'
ENTER '0' 'RETURN' TO END ENTRY.
ENTER NODE -->76
ENTER NODE -->79
ENTER NODE -->98
ENTER NODE -->118
ENTER NODE -->132
142
ENTER NODE -->142
ENTER NODE -->160
ENTER NODE -->174
ENTER NO1E -->
194
ENTER NODE -->208
ENTER NODE -->224
ENTER NODE -->238
ENTER NODE -->246
ENTER NODE -->268
ENTER NODE -->280
ENTER NODE -->301
ENTER NODE -->313
ENTER NODE -->322
ENTER NODE -->0

1 - RANDOM NODES
2 - RANGE OF NODES
3 - SPECIFIED PLANE
4 - ALL NODES
5 - EXIT

ENTER NODE SELECTION OPTION (1,2,...5) ---------->5

        40 BOUNDARY CONDITIONS ADDED
        40 BOUNDARY CONDITIONS TOTAL
```

```
:: STOP

LISTING OF DATA FILES AVAILABLE

$10 --- PROPS
$11 --- RENUMB4
$12 --- BOUNDS



     STOP
     064000 MAXIMUM EXECUTION FL.
     17.031 CP SECONDS EXECUTION TIME.
FILE QUOTA EXCEEDED
COMMAND- FILES
--LOCAL FILES--
 $INPUT     $OUTPUT   ZP         TAPE11     TAPE10
  TAPE14     TAPE12    TAPE13     TAPE17     TAPE16
  TAPE15     TAPE18
FILE QUOTA EXCEEDED
COMMAND- RETURN,TAPE10,TAPE11,TAPE13,TAPE14,TAPE15
COMMAND- RETURN,TAPE16,TAPE17,TAPE18
COMMAND- FILES
--LOCAL FILES--
 $INPUT     $OUTPUT   ZP         TAPE12
COMMAND- REWIND,TAPE12
COMMAND- COPYBF,TAPE12,UNFMT
 EOI ENCOUNTERED AFTER COPY OF FILE
          0, RECORD  1
COMMAND- RETURN,TAPE12
```

7.2.5 REFMT Execution

```
COMMAND- FILES
--LOCAL FILES--
 $INPUT    $OUTPUT   $P          UNFMT
COMMAND- BEGIN,REFMT,P.


 ****************************************

 BEGIN REFMT    --    MAGNA INPUT GENERATOR

 ****************************************


 ** SUMMARY OF INITIAL SCAN OF DATA FILE **

 -----------------------------------------

 NUMBER OF NODAL POINTS .......   336
 NUMBER OF ELEMENTS (TOTAL) ...    45
 NUMBER OF CONSTRAINT RECORDS..    40
 NUMBER OF LINEAR CONSTRAINTS..     0
 NUMBER OF NODAL LOADS ........     0
 NUMBER OF ELEMENT LOADS ......     0
 DISTINCT LOAD CASES / GROUPS..     0


     MAGNA        NUMBER OF     ELEMENTS WITH
 ELEMENT TYPE     ELEMENTS      UNSPEC. MATRL

       8             45              45


ENTER A THREE-LINE PROBLEM TITLE   (UP TO 80 CHARACTERS PER LINE)

.........1.........2.........3.........4.........5.........6.........7.........8
 REAR CANOPY SYMMETRIC MODEL - SIMPLE SUPPORTS - EIGENVALUE SOLUTION
 DATA GENERATED 18 JAN. 1983
 R.A.B.


********************************************

 MAJOR SOLUTION OPTIONS AND PARAMETERS

********************************************

ENTER ANALYSIS TYPE      (LINEAR,NONLINEAR) -LINEAR

ENTER ANALYSIS SUBTYPE  (STATIC,DYNAMIC)----DYNAMIC

DYNAMIC SOLUTION TYPE   (EIGEN ,TRANSIENT) -EIGEN

ENTER NUMBER OF FREQUENCIES TO BE COMPUTED -4

CONSISTENT MASS MATRICES TO BE USED (Y/N) --Y

POSTPROCESSOR FILE TO BE WRITTEN (Y/N) -----Y

********************************************
     END OF OPTIONS SPECIFICATIONS
********************************************


INDIVIDUAL ELEMENTS IN THE MODEL CONTAINUNDEFINED PROPERTIES

ELEMENT TYPE       =     8
NUMBER OF ELEMENTS=     45

PLEASE DEFINE A DEFAULT PROPERTY CODE FOR THIS ELEMENT TYPE,
OR ENTER MATERIALS DATA DIRECTLY BELOW.
```

```
MATERIAL PROPERTY DEFINITION OPTIONS

 E -- ENTER PROPERTY DATA DIRECTLY
 C -- SPECIFY A LIBRARY PROPERTY CODE
 L -- LIST SELECTED LIBRARY ENTRIES

ENTER OPTION  ( E , C , L ) -----------L

LIBRARY MATERIAL DESCRIPTIONS CAN BE LISTED BY MATERIAL TYPE
VALID MATERIAL TYPES ARE AS FOLLOWS ----

ACRYL - ACRYLICS
ALUMI - ALUMINUM ALLOYS
CASTI - CAST IRONS
COPPR - COPPER-BASED ALLOYS
GLASS - GLASSES
MAGNS - MAGNESIUM ALLOYS
NICKL - NICKEL ALLOYS
PLYMR - POLYMERIC MATERIALS
POLYC - POLYCARBONATES
STEEL - CARBON STEELS
STSTL - STAINLESS STEELS
TITNM - TITANIUM

ENTER MATERIAL TYPE (STEEL,STSTL,ETC.)--GLASS

MATL. CODE  ...........DESCRIPTION................

   00830    GLASS - SODA LIME GLASS, FULLY TEMPERED
   00835    GLASS - PPG HERCULITE-II, CHEM.STRENGTHENED
   00840    GLASS - ANNEALED GLASS, GENERIC
   00848    GLASS - PVB 0.20 PLASTICIZER, O.C.

MATERIAL PROPERTY DEFINITION OPTIONS

 E -- ENTER PROPERTY DATA DIRECTLY
 C -- SPECIFY A LIBRARY PROPERTY CODE
 L -- LIST SELECTED LIBRARY ENTRIES

ENTER OPTION  ( E , C , L ) -----------C

ENTER LIBRARY PROPERTY CODE --840


MATERIAL PROPERTIES DEFINITION FOR THE MODEL IS COMPLETE.
AT THIS POINT MATERIALS DATA MAY BE EDITED AS NECESSARY.
(NOTE THAT SOME DATA WHICH IS UNIMPORTANT FOR THE CURRENT
ANALYSIS MAY BE DEFINED AS ZERO)

CURRENT PROPERTIES ARE LISTED BELOW ----


CODE  MODULUS   POIS.RATIO  DENSITY   YIELD STR. THERM.EXP.

 -8  .1050E+08  .2200E+00  .2350E-03  .6900E+04 0.

L = (L)IST CURRENT PROPERTIES TABLE
C = (C)HANGE AN ENTRY IN THE TABLE
S = (S)TOP EDITING

ENTER OPTION  ( L , C , S ) -----------S


 ****************************************
 *                                      *
 *       DATA GENERATION COMPLETE       *
 *                                      *
 ****************************************


    STOP
    065500 MAXIMUM EXECUTION FL.
     1.063 CP SECONDS EXECUTION TIME.
COMMAND- REWIND,FDATA
COMMAND- REQUEST,DATA,*PF.
COMMAND- COPYBF,FDATA,DATA.
COMMAND- CATALOG,DATA,WINDSHIELDDATAEIGEN,ID=BROCKMAN,RP=200
```

7.3 EXECUTION OF IJKGEN / PREP / REFMT SEQUENCE

This Subsection contains a preprocessing example demonstrating the use of the default options provided in IJKGEN (Subsection 2.2) to generate a three-dimensional model of a folded plate-type structure. The use of user-written subroutines within IJKGEN is described separately in Appendix D.

The component to be modelled in this hypothetical example involves a folded plate containing both flat and cylindrical geometry. The plate is twenty inches wide, with symmetry assumed along its centerline. Its folded edges are represented by 90-degree cylindrical segments of mean radius 5.1 inches, and ten inches long (half the width of the plate). A flat segment between the folded edges is 40 inches in length, and symmetry is assumed along the center of this panel as well. The thickness of the entire panel is uniform, at 0.20 inches.

7.3.1 Summary of Modelling Procedure

The basic geometric components of the plate are the flat section and its cylindrical ends. These are generated in IJKGEN separately, using the default coordinate system and mesh grading options:

- Rectangular segment : Cartesian coordinates, graded mesh

- Cylindrical segment : Cylindrical coordinates, uniform mesh.

The rectangular (flat) portion of the mesh has been refined slightly near the symmetric center section, where a band of pressure will be applied in the analysis. The row of elements to be joined to the cylindrical end has been made four times as long as the row at the center, and the grid is uniform in the remaining two directions.

The two substructures created in IJKGEN are assembled with the use of PREP, after using the TRANslate and ROTAte operations to position them properly. Observe that the flat section of the plate could have been positioned properly in IJKGEN by an appropriate selection of the minima and maxima; however, the repositioning has been performed here to demonstrate more of the features of PREP.

The sequence of operations performed in PREP is summarized below for easy reference:

Step 1. Verification of IJKGEN-Generated Models

- PLOT	(Plot original cylinder data)
- MASK	(Convert cylinder to 16-node elements)
- PLOT	(Plot original plate data)
- MASK	(Convert plate to 16-node elements)

Step 2. Repositioning of Plate Model Segment

- ROTAte	(Two 90-degree rotations of plate model)

- PLOT (Verify that rotations are correct)
- TRANslate (Move plate next to cylindrical segment)
- PLOT (Verify that plate is positioned)

Step 3. Combination of Flat Plate and Cylindrical Sections

- MERGe (Combine the two models)
- PLOT (Verify that it has been done correctly)

Step 4. Finish Geometric Model

- RENUmber (Reorder nodes to reduce bandwidth)
- RENUmber (Verify that a second pass not needed)

Step 5. Apply Boundary Conditions and Loads

- BOUNds (Apply symmetry conditions on the edges Z=Ø and Y=-1Ø.; clamp the free end of the cylindrical segment, X=Ø.)
- PLOT (Use ORIENT plot option to determine the correct surface numbers for pressures)
- LOADs (Apply inward pressure on the upper faces of elements 41 through 5Ø)

Step 6. Finish Model by Assigning Properties

- PROPerty (Assign all elements as solids, to be integrated using a 14-point quadrature rule; leave the specification of materials data for the REFMT step)

- LIST (Make sure of the local file number which contains the completed model; in this case, local file TAPE12 must be saved)
- STOP (Leave PREP)

The above sequence of operations is typical for modelling data generated with IJKGEN, in which the original model can be generated with the necessary degree of refinement. It is worth noting that the specification of properties, which is quite simple in this case, can become more tedious when several materials are involved and a number of REFIne operations are needed. In such cases, it is a good practice to assign the properties at an early stage, where the element numbering sequence is generally more regular.

The final phase of model generation is the REFMT step, in which a complete data deck for the MAGNA program is generated in this example. The initial output from REFMT indicates that the data has been read correctly, and that REFMT has properly interpreted the 16-node elements as being Type 8 elements in MAGNA. The following options have been selected through additional user inputs:

- nonlinear, static solution;

- 10-increment solution, using a constant time step;

- combined full and modified Newton-Raphson iterations;

- printing at every other increment, and writing of a post-processor file at every five increments;

- restart file to be written at every other increment, with the restart file label FPLT.

The next segment of REFMT execution deals with material properties, since none were specified within PREP. Since al elements in the model contain the same material code (zero), the user is requested to define a default material code for all Type 8 elements having this material code. Entries for carbon steels are listed from the materials library, and a default material code of 120 (G10500 steel) is selected. No editing of the materials data is performed, although entries may be altered selectively as many times as necessary.

Finally, information is requested concerning the variation of the applied loads for the nonlinear solution. Since the solution is static, a ramp function is selected (option 3), causing the loads to applied incrementally in equal steps. The pressure magnitude input in PREP is retained as the peak loading value, and all pressures are defined as being 'live loads', to be updated according to the current surface orientation continuously during the nonlinear analysis.

Once the data generation is complete, the local file FDATA, which contains the complete input deck for MAGNA, is cataloged as a permanent disk file with the name FOLDEDPLATEDATA. All that remains to perform the nonlinear analysis is the submission of the proper job control stream for MAGNA, as described in Reference [4].

The complete modelling session, including all printed output and plotting, is reproduced in Paragraphs 7.3.2 through 7.3.4 below.

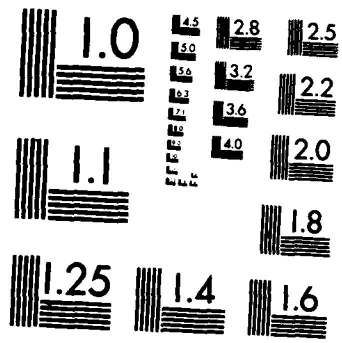
7.3.2 IJKGEN Execution

```
COMMAND- ATTACH,P,PREPROCESSORPROC,ID=BROCKMAN,MR=1.
 AT CY= 998 SN=AFFDL
COMMAND- BEGIN,IJKGEN,P.


          ****************************************
          **********-    BEGIN IJKGEN    **********
          ****************************************

 I J K G E N   -   GENERATION OF GEOMETRIC MESH DATA FOR SOLID,
 THICK SHELL OR PLATE FINITE ELEMENT MODELS, USING AN INTEGER -
 COORDINATE INDEXING SCHEME.  OPTIONAL USER ROUTINES ARE -
   (1)  SURFAC (I,J,K,ALPHA,BETA,ZETA) - DEFINE MESH GEOMETRY
   (2)  CRDTRN (ALPHA,BETA,ZETA,X,Y,Z) - COORD. TRANSFORMATION
 BUILT-IN OPTIONS INCLUDE RECTANGULAR, CYLINDRICAL OR SPHERICAL
 COORDINATES, AND UNIFORM OR PROPORTIONALLY GRADED MESH SPACING

   (3)  UINPUT - USER PARAMETER INPUT ROUTINE (INITIALIZE DATA
                 IN BLANK COMMON)

 ..............................................................

 **********  USER SUBROUTINE 'SURFAC' NOT GIVEN  **********

 BUILT-IN MESH DIVISION OPTIONS ARE AS FOLLOWS -
  (1) - UNIFORM MESH IN EACH DIRECTION
  (2) - GRADED MESH (SPECIFY RATIO OF FIRST/LAST ELEMENT SIZE

 ENTER OPTION ( 1 , 2 ) ................2
 ENTER THE RATIO OF FIRST / LAST ELEMENT LENGTHS FOR EACH
 COORDINATE DIRECTION (ALPHA, BETA, ZETA) (R=1 FOR UNIFORM)

 ENTER LENGTH RATIOS (R1,R2,R3) ........ 4., 1., 1.

 **********- USER SUBROUTINE 'CRDTRN' NOT GIVEN  **********

 BUILT-IN COORDINATE SYSTEM TRANSFORMATION OPTIONS ARE - - -
 (1) RECTANGULAR,   (2) CYLINDRICAL,   (3) SPHERICAL

 ENTER COORDINATE SYSTEM OPTION (1,2,3) -1

 ** PLEASE NOTE THE FOLLOWING CONVENTIONS FOR RECTANGULAR SYSTEM **

  'ALPHA' = X      'BETA' = Y      'ZETA' = Z
 A RIGHT-HANDED SYSTEM IS ASSUMED.

 ENTER LIMITING SURFACE COORDINATE VALUES -

  1. ALPHA(MIN)     2. ALPHA(MAX)
  3. BETA (MIN)     4. BETA (MAX)
  5. ZETA (MIN)     6. ZETA (MAX)

0.0, 20.0, 0.0, 10., 0.0, 0.2


 ENTER THE NUMBER OF ELEMENTS TO BE GEN-
  ERATED IN THE ALPHA, BETA AND ZETA CO-
  ORDINATE DIRECTIONS, RESPECTIVELY .... 6, 5, 1


 *** BEGIN GENERATION PHASE ***

 NUMBER OF NODES TO BE GENERATED   =  420
 NUMBER OF ELEMENTS TO BE GENERATED=   30

 THE THICKNESS DIRECTION OF THE MODEL (IF ONE EXISTS, AS IN A
 THICK SHELL) MUST BE IDENTIFIED TO ORIENT ELEMENTS PROPERLY.

 OPTIONS ARE (1)ALPHA, (2)BETA, (3)ZETA, OR (4)&lt;UNIMPORTANT&gt;.

 ENTER THICKNESS DIRECTION CODE (1,2,3,4) -3


 *** DATA GENERATION COMPLETE ***

 ****** IJKGEN  TERMINATED ******
     STOP
     006100 MAXIMUM EXECUTION FL.
      0.179 CP SECONDS EXECUTION TIME.
COMMAND- REWIND,UNFMT
COMMAND- COPYBF,UNFMT,TAPE11
 EOI ENCOUNTERED AFTER COPY OF FILE
         0, RECORD  1
COMMAND- RETURN,UNFMT.
```

```
COMMAND- BEGIN,IJKGEN,P.


          *****************************************
          **********     BEGIN IJKGEN     **********
          *****************************************

I J K G E N   -   GENERATION OF GEOMETRIC MESH DATA FOR SOLID,
THICK SHELL OR PLATE FINITE ELEMENT MODELS, USING AN INTEGER -
COORDINATE INDEXING SCHEME.  OPTIONAL USER ROUTINES ARE -
  (1)  SURFAC (I,J,K,ALPHA,BETA,ZETA) - DEFINE MESH GEOMETRY
  (2)  CRDTRN (ALPHA,BETA,ZETA,X,Y,Z) - COORD. TRANSFORMATION
BUILT-IN OPTIONS INCLUDE RECTANGULAR, CYLINDRICAL OR SPHERICAL
COORDINATES, AND UNIFORM OR PROPORTIONALLY GRADED MESH SPACING

  (3)  UINPUT - USER PARAMETER INPUT ROUTINE (INITIALIZE DATA
              IN BLANK COMMON)

..............................................................

**********  USER SUBROUTINE 'SURFAC' NOT GIVEN  **********

BUILT-IN MESH DIVISION OPTIONS ARE AS FOLLOWS -
 (1) - UNIFORM MESH IN EACH DIRECTION
 (2) - GRADED MESH (SPECIFY RATIO OF FIRST/LAST ELEMENT SIZE

ENTER OPTION ( 1 , 2 ) ................1

**********  USER SUBROUTINE 'CRDTRN' NOT GIVEN  **********

BUILT-IN COORDINATE SYSTEM TRANSFORMATION OPTIONS ARE - - -
(1) RECTANGULAR,  (2) CYLINDRICAL,  (3) SPHERICAL

ENTER COORDINATE SYSTEM OPTION (1,2,3) -2

** PLEASE NOTE THE FOLLOWING CONVENTIONS FOR CYLINDRICAL SYSTEM **

 'ALPHA' = RADIUS      'BETA' = ANGLE      'ZETA' = AXIAL
SYSTEM IS RIGHT-HANDED, WITH ALL ANGLES MEASURED IN DEGREES.

ENTER LIMITING SURFACE COORDINATE VALUES -

 1. ALPHA(MIN)    2. ALPHA(MAX)
 3. BETA (MIN)    4. BETA (MAX)
 5. ZETA (MIN)    6. ZETA (MAX)

5.0, 5.2, 0.0, 90.0, 0.0, 10.0


ENTER THE NUMBER OF ELEMENTS TO BE GEN-
  ERATED IN THE ALPHA, BETA AND ZETA CO-
  ORDINATE DIRECTIONS, RESPECTIVELY .... 1,4,5

*** BEGIN GENERATION PHASE ***

NUMBER OF NODES TO BE GENERATED    =  297
NUMBER OF ELEMENTS TO BE GENERATED=   20

THE THICKNESS DIRECTION OF THE MODEL (IF ONE EXISTS, AS IN A
THICK SHELL) MUST BE IDENTIFIED TO ORIENT ELEMENTS PROPERLY.

OPTIONS ARE (1)ALPHA, (2)BETA, (3)ZETA, OR (4)<UNIMPORTANT>.

ENTER THICKNESS DIRECTION CODE (1,2,3,4) - 1


*** DATA GENERATION COMPLETE ***

****** IJKGEN  TERMINATED ******
     STOP
     026100 MAXIMUM EXECUTION FL.
      0.141 CP SECONDS EXECUTION TIME.
COMMAND- REWIND,UNFMT.
COMMAND- COPYBF,UNFMT,TAPE10.
 EOI ENCOUNTERED AFTER COPY OF FILE
          0, RECORD  1
COMMAND- RETURN,UNFMT.
```

AD-A129 825 MAGNA (MATERIALLY AND GEOMETRICALLY NONLINEAR ANALYSIS) 3/3
PART II PREPROCES..(U) DAYTON UNIV OH RESEARCH INST
T S BRUNER ET AL. DEC 82 UDR-TR-82-112
UNCLASSIFIED AFWAL-TR-82-3098-PT-2 F33615-80-C-3403 F/G 9/2 NL

END
FILMED
DTIC

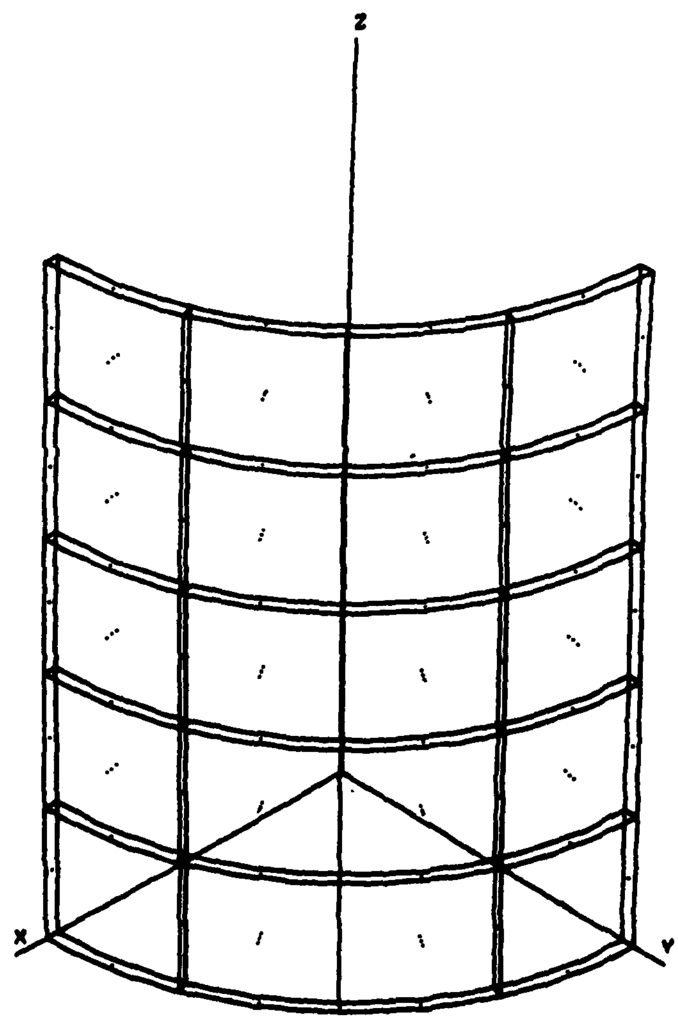


MICROCOPY RESOLUTION TEST CHART
NATIONAL BUREAU OF STANDARDS-1963-A

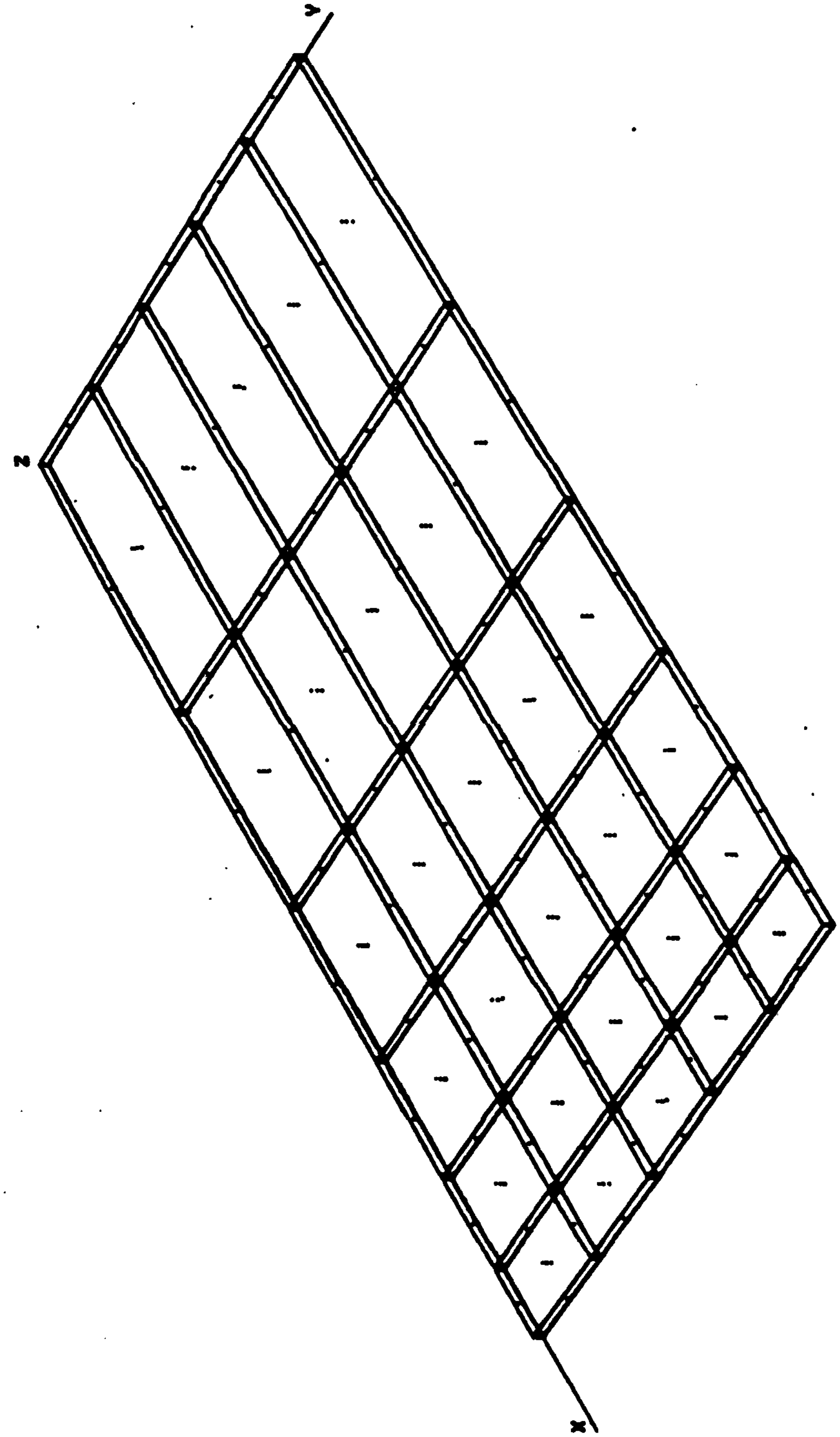
7.3.3 PREP Execution

```
COMMAND- BEGIN,PREP,P.

        ********************************************
        *************   BEGIN PREP   ************Tu
        **** MAGNA PREPROCESSING UTILITIES *****
        ********************************************

PLOTTING TERMINAL ( Y , N ) ?;;Y
TEKTRONIX OR HP ( T , H ) ?  ;;T

TEKTRONIX TERMINAL TYPES ----
 0. 4006-1
 1. 4010 / 4012 / 4013 / 4052
 2. 4014 / 4015
 3. 4014 / 4015    ( ENH. GR. MOD. )
 4. 4114

ENTER TERMINAL TYPE ( 0 - 4 );;3
ENTER CHARACTERS PER SECOND  ;;120
NUMBER OF INPUT DATA FILES ? ;;2
ENTER FILE $                 ;;10
ENTER LABEL                  ;;CYL-1
ENTER FILE $                 ;;11
ENTER LABEL                  ;;PLT-1

TYPE 'HELP' FOR LIST OF COMMANDS

:: LIST

LISTING OF DATA FILES AVAILABLE

$10 --- CYL-1
$11 --- PLT-1

:: PLOT
ENTER INPUT FILE $1 LABEL                  ;;CYL-1

INITIALIZATION OF PLOTTING ROUTINE ERASES SCREEN.
READY (Y,N) ? ....: Y
```

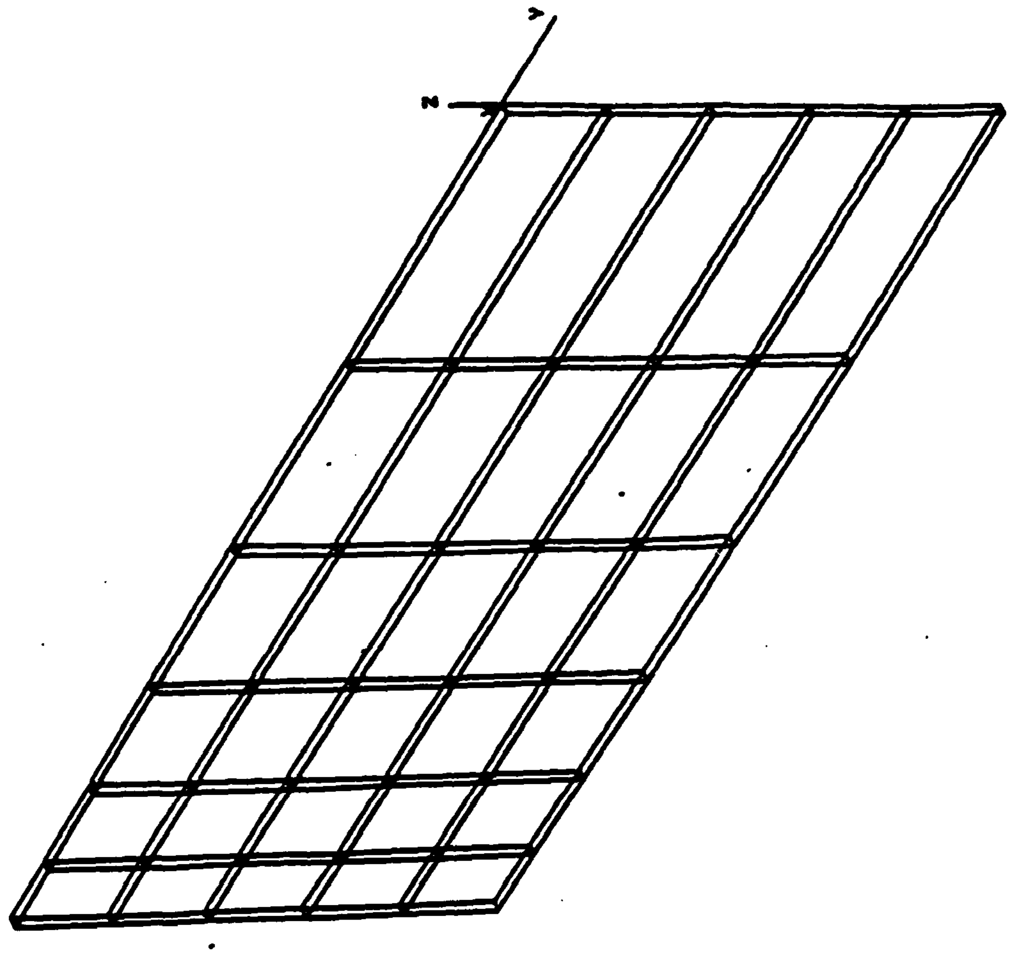
```
$AXES
  PLOT AND LABEL THE AXES?(Y,N) ...............: Y
$DRAW

 AT END OF PLOT, ENTER CHARACTER TO CONTINUE.
 READY (Y,N).? ....: Y
```



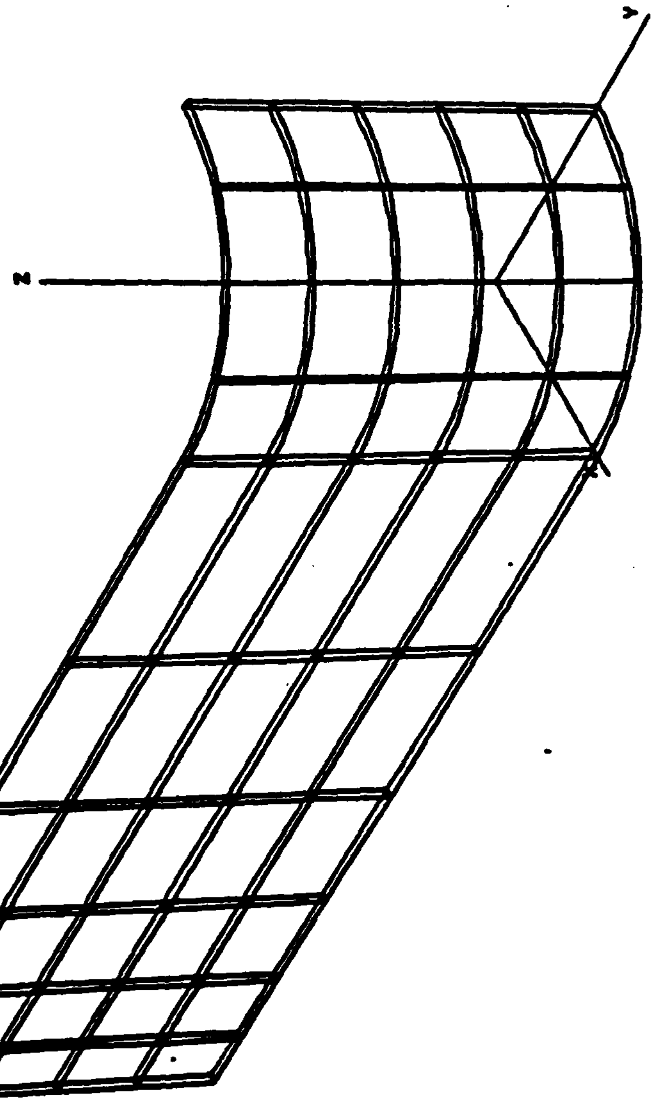
```
?MAIN

:: MASK
ENTER INPUT FILE #1 LABEL                    ;;CYL-1
ENTER OUTPUT FILE LABEL                      ;;CYL-16N
CREATING NEW DATA FILE
MASK OPTIONS -
  1. 3-D ELEMENTS ONLY
  2. 2-D ELEMENTS ONLY
ENTER OPTION...;;1
MAX NODE POSITION # IN FINAL ELEMENTS?
  (8,16,20,26,27) ;;16

MASK COMPLETE.
PERFORM SUBSEQUENT SIFT ? (Y,N) ->Y

      SIFT COMPLETE
  130 UNUSED NODES DELETED

:: LIST

LISTING OF DATA FILES AVAILABLE

810 --- CYL-1
811 --- PLT-1
812 --- CYL-16N

:: PLOT
ENTER INPUT FILE #1 LABEL                    ;;PLT-1
 IDRAW

  AT END OF PLOT, ENTER CHARACTER TO CONTINUE.
  READY (Y,N) ? ....: Y
```


Y
Z
X

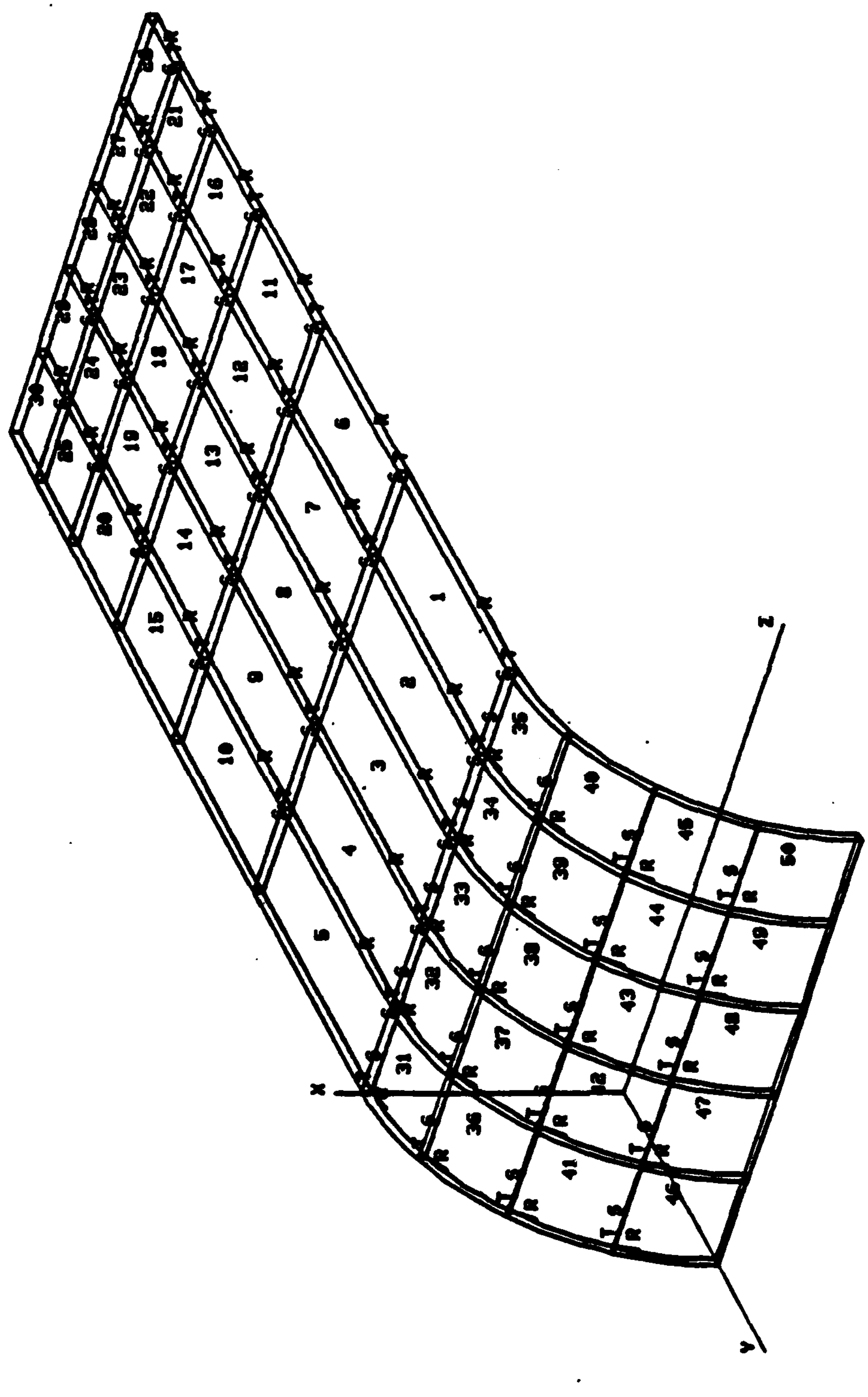
```
XMAIN

:: MASK
ENTER INPUT FILE #1 LABEL                   ;;PLT-1
ENTER OUTPUT FILE LABEL                     ;;PLT-16N
CREATING NEW DATA FILE
MASK OPTIONS -
  1. 3-D ELEMENTS ONLY
  2. 2-D ELEMENTS ONLY
ENTER OPTION...;;1
MAX NODE POSITION # IN FINAL ELEMENTS?
  (8,16,20,26,27) ;;16

MASK COMPLETE.
PERFORM SUBSEQUENT SIFT ? (Y,N) ->Y

      SIFT COMPLETE
  203 UNUSED NODES DELETED

:: LIST

LISTING OF DATA FILES AVAILABLE

#10 --- CYL-1
#11 --- PLT-1
#12 --- CYL-16N
#13 --- PLT-16N

:: DELETE
ENTER LABEL              ;;CYL-1

:: DELETE
ENTER LABEL              ;;PLT-1

:: ROTATE
ENTER INPUT FILE #1 LABEL                   ;;PLT-16N
ENTER OUTPUT FILE LABEL                     ;;PLT-ROT
CREATING NEW DATA FILE
ROTATIONS PERFORMED ABOUT X, THEN Y, THEN Z.
ENTER ROTATION ANGLES IN DEGREES (X,Y,Z) -->-90., 0.0, -90.

:: LIST

LISTING OF DATA FILES AVAILABLE

#10 --- PLT-ROT
#12 --- CYL-16N
#13 --- PLT-16N

:: PLOT
ENTER INPUT FILE #1 LABEL                   ;;PLT-ROT
 XDRAW

  AT END OF PLOT, ENTER CHARACTER TO CONTINUE.
  READY (Y,N) ? ....: Y
```


Y
Z

```
%MAIN

:: LIST

LISTING OF DATA FILES AVAILABLE

#10 --- PLT-ROT
#12 --- CYL-16N
#13 --- PLT-16N

:: TRANSLATE
ENTER INPUT FILE #1 LABEL                    ;;PLT-ROT
ENTER OUTPUT FILE LABEL                      ;;PLT-RDY
CREATING NEW DATA FILE
ENTER TRANSLATION FACTORS(XFAC,YFAC,ZFAC)-----: 500, 0.0, 10.0

:: DELETE
ENTER LABEL                    ;;PLT-ROT

:: MERGE
ENTER INPUT FILE #1 LABEL                    ;;PLT-RDY
ENTER INPUT FILE #2 LABEL                    ;;CYL-16N
ENTER OUTPUT FILE LABEL                      ;;BOTH
CREATING NEW DATA FILE

TIDY OPTION ONE OR TWO ? (OPT1, OPT2, HELP) ->OPT2

      TIDY COMPLETE
   11 DUPLICATE NODES DELETED

:: PLOT
ENTER INPUT FILE #1 LABEL                    ;;BOTH
 %DRAW

  AT END OF PLOT, ENTER CHARACTER TO CONTINUE.
  READY (Y,N) ? ....: Y
```



```
$MAIN

:: LIST

LISTING OF DATA FILES AVAILABLE

$10 --- BOTH
$11 --- PLT-RDY
$12 --- CYL-1GN
$13 --- PLT-1GN

:: DELETE
ENTER LABEL                    ;;PLT-RDY

:: DELETE
ENTER LABEL                    ;;CYL-1GN

:: DELETE
ENTER LABEL                    ;;PLT-1GN)

:: LIST

LISTING OF DATA FILES AVAILABLE

$10 --- BOTH

:: PLOT
ENTER INPUT FILE $1 LABEL                ;;BOTH
 $VERT
   WHICH AXIS IS VERTICAL?
   ENTER 1 FOR X, 2 FOR Y, OR 3 FOR Z ..........: 1
 $EYE
   ENTER THE EYE POSITION,
   XEYE, YEYE, ZEYE ............................: 300 500 400
 $LABE
   LABEL THE ELEMENTS?(Y,N) ....................: Y
   LABEL THE NODES?(Y,N) .......................: N
 $ORIENT
   PLOT ORIENTATION AXES?(Y,N) .................: Y
 $DRAW

  AT END OF PLOT, ENTER CHARACTER TO CONTINUE.
  READY (Y,N) ? ....: Y
```



```
%MAIN

:: LIST

LISTING OF DATA FILES AVAILABLE

810 --- BOTH

:: RENUMBER
ENTER INPUT FILE 81 LABEL                   ;;BOTH
ENTER OUTPUT FILE LABEL                     ;;BOTH/R
CREATING NEW DATA FILE

 NODE POINT REORDERING COMPLETE

 NODAL BANDWIDTH (OLD) =   59
 NODAL BANDWIDTH (NEW) =   59

:: DELE
ENTER LABEL                    ;;BOTH/R

:: BOUNDS
ENTER INPUT FILE 81 LABEL                   ;;BOTH
ENTER OUTPUT FILE LABEL                     ;;GEOM + BCS
CREATING NEW DATA FILE

BOUNDARY CONDITIONS  MAY  BE ENTERED  FOR  RANDOM
  NODES,  RANGES OF NODES,  ALL NODES ON A PLANE,
  OR THE ENTIRE MODEL.

FOR EACH GROUP OF NODES,  CONSTRAINTS  MAY BE SET
  IN ANY COMBINATION OF THE X, Y, & Z DIRECTIONS.
  (NOTE THAT 2-D MODELS MUST BE FIXED IN Z)

CONSTRAINT DIRECTION CODE CONSTRUCTED AS FOLLOWS:

  ENTRY 1 - '1' FOR CONSTRAINED IN X DIRECTION
            '0' FOR UNCONSTRAINED IN X DIRECTION
  ENTRY 2 -     SAME FOR Y
  ENTRY 3 -     SAME FOR Z

  EXAMPLE -> '1,0,1' FOR CONSTRAINTS IN X AND Z
                     DIRECTIONS BUT NOT IN Y.

1 - RANDOM NODES
2 - RANGE OF NODES
3 - SPECIFIED PLANE
4 - ALL NODES
5 - EXIT

ENTER NODE SELECTION OPTION (1,2,...5) --------->3

ENTER CONSTRAINT DIRECTION CODE (THREE VALUES) ->0 0 1

PLANE IS DEFINED BY  AX + BY + CZ = D.
  ENTER COEFFICIENTS ( A , B , C , D) --------->0 0 1 0

             43 NODES FOUND

 1 - RANDOM NODES
 2 - RANGE OF NODES
 3 - SPECIFIED PLANE
 4 - ALL NODES
 5 - EXIT

 ENTER NODE SELECTION OPTION (1,2,...5) --------->3

 ENTER CONSTRAINT DIRECTION CODE (THREE VALUES) ->1 1 0

 PLANE IS DEFINED BY  AX + BY + CZ = D.
   ENTER COEFFICIENTS ( A , B , C , D) --------->1 0 0 0

             22 NODES FOUND

 1 - RANDOM NODES
 2 - RANGE OF NODES
 3 - SPECIFIED PLANE
 4 - ALL NODES
 5 - EXIT
```

```
ENTER NODE SELECTION OPTION (1,2,...5) --------->3

ENTER CONSTRAINT DIRECTION CODE (THREE VALUES) ->0 1 0

PLANE IS DEFINED BY  AX + BY + CZ = D.
  ENTER COEFFICIENTS ( A , B , C , D) ---------->0 1 0 -10.

                 0 NODES FOUND

1 - RANDOM NODES
2 - RANGE OF NODES
3 - SPECIFIED PLANE
4 - ALL NODES
5 - EXIT

ENTER NODE SELECTION OPTION (1,2,...5) --------->3

ENTER CONSTRAINT DIRECTION CODE (THREE VALUES) ->0 1 0

PLANE IS DEFINED BY  AX + BY + CZ = D.
  ENTER COEFFICIENTS ( A , B , C , D) ---------->0 1 0 -20

                22 NODES FOUND

1 - RANDOM NODES
2 - RANGE OF NODES
3 - SPECIFIED PLANE
4 - ALL NODES
5 - EXIT

ENTER NODE SELECTION OPTION (1,2,...5) --------->5


                87 BOUNDARY CONDITIONS ADDED
                87 BOUNDARY CONDITIONS TOTAL

:: LIST

LISTING OF DATA FILES AVAILABLE

#10 --- BOTH
#11 --- GEOM + BCS

:: DELETE
BOTH
ENTER LABEL                     ::

:: LOAD
ENTER INPUT FILE #1 LABEL                    ;;GEOM + BCS
ENTER OUTPUT FILE LABEL                      ;;ALL BUT PROPS
CREATING NEW DATA FILE

LOAD SPECIFICATION OPTIONS
  N - NODAL LOAD ENTRY
  E - ELEMENT LOAD ENTRY
  L - LIST EXISTING LOADS
  H - PRINTS THIS LIST
  S - STOP LOAD ENTRY

ENTER LOAD SPECIFICATION OPTION    --->E



ELEMENT SPECIFICATION OPTIONS
  A - ALL ELEMENTS
  S - SINGLE ELEMENT
  R - RANGE OF ELEMENTS
  H - PRINTS THIS LIST
  E - EXIT ELEMENT LOAD SPECIFICATION SECTION

ENTER ELEMENT SPECIFICATION OPTION --->

ENTER ELEMENT SPECIFICATION OPTION --->R
ENTER CASE NUMBER                  --->1
ENTER SURFACE NUMBER               --->6
ENTER PRESSURE                     --->-1.0
ENTER BEGINNING ELEMENT,
      ENDING ELEMENT,
      AND INCREMENT                --->21,30,1
```

```
ENTER ELEMENT SPECIFICATION OPTION --->E

LEAVING ELEMENT LOAD SPECIFICATION SECTION

ENTER LOAD SPECIFICATION OPTION    --->S

LEAVING LOAD SPECIFICATION MODULE

:: LIST

LISTING OF DATA FILES AVAILABLE

$10 --- ALL BUT PROPS
$11 --- GEOM + BCS

:: DELETE
ENTER LABEL                    ;;GEOM + BCS

:: TIME
   THE CPU TIME FROM THE START OF THIS SESSION IS    13.031 SEC.

:: PROPS
ENTER INPUT FILE $1 LABEL                ;;ALL BUT PROPS
ENTER OUTPUT FILE LABEL                  ;;FINAL
CREATING NEW DATA FILE

ELEMENT INPUT OPTIONS
  1)  ALL 2-D ELEMENTS
  2)  ALL 3-D ELEMENTS
  3)  ALL ELEMENTS
  4)  RANDOM ELEMENTS
  5)  RANGE OF ELEMENTS
ENTER OPTION...: 3

ELEMENT TYPES AVAILABLE...
   0 - DEFAULT (BAR,MEMBRANE,SOLID)
   1 - SHELL OR BEAM
   2 - PLANE STRESS (MEMBRANE)
   3 - PLANE STRAIN
   4 - AXISYMMETRIC
   5 - SHEAR PANEL
   6 - CONTACT ELEMENT
PRESENT ELEMENT TYPE =    0
CHANGE (Y,N) ..........: N

MATERIAL TYPE =    0
CHANGE (Y,N) ..........: N

INTEGRATION OPTIONS AVAILABLE -
   0 - DEFAULT FOR ELEMENT TYPE
   1 - SINGLE POINT
   6 - MIDFACE POINTS
   8 - 2 - PT. GAUSS
  14 - IRON'S RULE
  27 - 3 - PT. GAUSS
PRESENT INTEGRATION ORDER =    0
CHANGE (Y,N) ..........: Y
ENTER NEW INTEGRATION ORDER...: 14

ELEMENT PRINTING OPTIONS ----
   0 - PRINTING ON
   1 - PRINTING OFF

PRESENT PRINTING OPTION =  0
CHANGE ( Y, N ) ......: N
```

```
:: STOP

LISTING OF DATA FILES AVAILABLE

$10 --- ALL BUT PROPS
$11 --- FINAL

     STOP
     064100 MAXIMUM EXECUTION FL.
     10.870 CP SECONDS EXECUTION TIME.
FILE QUOTA EXCEEDED
COMMAND- FILES
--LOCAL FILES--
  TAPE18    TAPE10   $INPUT    $OUTPUT    ZP
  TAPE11    TAPE12    TAPE15    TAPE13    TAPE14
  TAPE16    TAPE17
FILE QUOTA EXCEEDED
COMMAND- RETURN,TAPE10,TAPE12,TAPE13,TAPE14,TAPE15,TAPE16,TAPE17
COMMAND- FILES
--LOCAL FILES--
  TAPE18   $INPUT    $OUTPUT   ZP         TAPE11
COMMAND- RETURN,TAPE18
COMMAND- REWIND,TAPE11
COMMAND- COPYBF,TAPE11,UNFMT
 EOI ENCOUNTERED AFTER COPY OF FILE
          0, RECORD  1
COMMAND- RETURN,TAPE11
```

7.3.4 REFMT Execution

```
COMMAND- BEGIN.REFMT,P.


 ****************************************
 BEGIN REFMT    --    MAGNA INPUT GENERATOR
 ****************************************


 ** SUMMARY OF INITIAL SCAN OF DATA FILE **
 -----------------------------------------

 NUMBER OF NODAL POINTS .......   373
 NUMBER OF ELEMENTS (TOTAL) ...    50
 NUMBER OF CONSTRAINT RECORDS..    87
 NUMBER OF LINEAR CONSTRAINTS..     0
 NUMBER OF NODAL LOADS ........     0
 NUMBER OF ELEMENT LOADS ......     1
 DISTINCT LOAD CASES / GROUPS..     1


     MAGNA         NUMBER OF     ELEMENTS WITH
 ELEMENT TYPE      ELEMENTS     UNSPEC. MATRL

       8              50             50

ENTER A THREE-LINE PROBLEM TITLE   (UP TO 80 CHARACTERS PER LINE)

.........1.........2.........3.........4.........5.........6.........7.........8
     FOLDED PLATE SEGMENT, DOUBLY SYMMETRIC, PRESSURE BAND AT CENTER
                 NONLINEAR STATIC ANALYSIS WITH MAGNA


 ****************************************
  MAJOR SOLUTION OPTIONS AND PARAMETERS
 ****************************************

 ENTER ANALYSIS TYPE      (LINEAR,NONLINEAR) -NONLINEAR

 ENTER ANALYSIS SUBTYPE  (STATIC,DYNAMIC)----STATIC

 ENTER TIME STEP OPTION  (CONST.,VARIABLE) --CONST

 ENTER THE INITIAL SOLUTION TIME STEP --------10.

 ENTER THE NUMBER OF SOLUTION TIME STEPS ----10

 ENTER PRINTING FREQUENCY, IN INCREMENTS ----2

 EQUILIBRIUM ITERATION OPTIONS ARE AS FOLLOWS

     0  =  NO ITERATION
     1  =  MODIFIED NEWTON (CONST. STIFFNESS)
     2  =  FULL NEWTON-RAPHSON ITERATION
     3  =  COMBINED FULL/MODIFIED NEWTON

 ENTER ITERATIVE SOLUTION OPTION (0,1,2,3) --3

 POSTPROCESSOR FILE TO BE WRITTEN (Y/N) ------Y

 ENTER THE FREQUENCY (IN INCREMENTS) AT WHICH
  RESULTS ARE TO SAVED ON POSTPROC. FILE ----5

 ARE RESTART FILES TO BE READ     (Y/N) ------N

 ARE RESTART FILES TO BE WRITTEN (Y/N) ------Y

 ENTER NEW RESTART FILE LABEL (4 CHARS.) ----FPLT
```

```
ENTER THE NUMBER OF INCREMENTS BETWEEN
 CHECKPOINTS ON THE NEW RESTART FILE -------2

*******************************************
     END OF OPTIONS SPECIFICATIONS
*******************************************


INDIVIDUAL ELEMENTS IN THE MODEL CONTAINUNDEFINED PROPERTIES

ELEMENT TYPE       =     8
NUMBER OF ELEMENTS=    50

PLEASE DEFINE A DEFAULT PROPERTY CODE FOR THIS ELEMENT TYPE,
OR ENTER MATERIALS DATA DIRECTLY BELOW.

MATERIAL PROPERTY DEFINITION OPTIONS

 E -- ENTER PROPERTY DATA DIRECTLY
 C -- SPECIFY A LIBRARY PROPERTY CODE
 L -- LIST SELECTED LIBRARY ENTRIES

ENTER OPTION  ( E , C , L ) -----------L

LIBRARY MATERIAL DESCRIPTIONS CAN BE LISTED BY MATERIAL TYPE
VALID MATERIAL TYPES ARE AS FOLLOWS ----

ACRYL - ACRYLICS
ALUMI - ALUMINUM ALLOYS
CASTI - CAST IRONS
COPPR - COPPER-BASED ALLOYS
GLASS - GLASSES
MAGNS - MAGNESIUM ALLOYS
NICKL - NICKEL ALLOYS
PLYMR - POLYMERIC MATERIALS
POLYC - POLYCARBONATES
STEEL - CARBON STEELS
STSTL - STAINLESS STEELS
TITNM - TITANIUM

ENTER MATERIAL TYPE (STEEL,STSTL,ETC.)--STEEL

MATL. CODE  ...........DESCRIPTION................

   00100     STEEL - UNS-G 10100   0.1 C    HR
   00101     STEEL - UNS-G 10100   0.1 C    CD
   00110     STEEL - UNS-G 10350   .35 C    HR
   00111     STEEL - UNS-G 10350   .35 C    CD
   00112     STEEL - UNS-G 10350   .35 C    DRAWN 800.F
   00120     STEEL - UNS-G 10500   0.5 C    HR
   00121     STEEL - UNS-G 10500   0.5 C    CD
   00122     STEEL - UNS-G 10500   0.5 C    DRAWN 900.F
   00150     STEEL - UNS-G 41400   0.4C-CR-MO   HR
   00151     STEEL - UNS-G 41400   0.4C-CR-MO   CD

MATERIAL PROPERTY DEFINITION OPTIONS

 E -- ENTER PROPERTY DATA DIRECTLY
 C -- SPECIFY A LIBRARY PROPERTY CODE
 L -- LIST SELECTED LIBRARY ENTRIES

ENTER OPTION  ( E , C , L ) -----------C

ENTER LIBRARY PROPERTY CODE --150

MATERIAL PROPERTIES DEFINITION FOR THE MODEL IS COMPLETE.
AT THIS POINT MATERIALS DATA MAY BE EDITED AS NECESSARY.
(NOTE THAT SOME DATA WHICH IS UNIMPORTANT FOR THE CURRENT
ANALYSIS MAY BE DEFINED AS ZERO)

CURRENT PROPERTIES ARE LISTED BELOW ----
```

```
 CODE  MODULUS   POIS.RATIO  DENSITY   YIELD STR. THERM.EXP.

  -8  .3000E+08  .2910E+00  .7250E-03  .6300E+05  .7300E-05

L = (L)IST CURRENT PROPERTIES TABLE
C = (C)HANGE AN ENTRY IN THE TABLE
S = (S)TOP EDITING

ENTER OPTION  ( L , C , S ) -----------C
(M)ODULUS            (P)OISSONS RATIO
(D)ENSITY            (Y)IELD STRESS
(T)HERMAL EXP. COEFF.

ENTER QUANTITY TO BE CHANGED (M,P,D,Y,T)---P

ENTER PROPERTY CODE (AS SHOWN IN TABLE)- -8

ENTER POISSONS RATIO ----------------- 0.28

L = (L)IST CURRENT PROPERTIES TABLE
C = (C)HANGE AN ENTRY IN THE TABLE
S = (S)TOP EDITING

ENTER OPTION  ( L , C , S ) -----------L

 CODE  MODULUS   POIS.RATIO  DENSITY   YIELD STR. THERM.EXP.

  -8  .3000E+08  .2800E+00  .7250E-03  .6300E+05  .7300E-05

L = (L)IST CURRENT PROPERTIES TABLE
C = (C)HANGE AN ENTRY IN THE TABLE
S = (S)TOP EDITING

ENTER OPTION  ( L , C , S ) -----------C
(M)ODULUS            (P)OISSONS RATIO
(D)ENSITY            (Y)IELD STRESS
(T)HERMAL EXP. COEFF.

ENTER QUANTITY TO BE CHANGED (M,P,D,Y,T)---Y

ENTER PROPERTY CODE (AS SHOWN IN TABLE)- -8

ENTER YIELD STRESS ------------------- 60000.

L = (L)IST CURRENT PROPERTIES TABLE
C = (C)HANGE AN ENTRY IN THE TABLE
S = (S)TOP EDITING

ENTER OPTION  ( L , C , S ) -----------S



NUMBER OF TIME FUNCTIONS TO BE GENERATED =     1

THESE TIME FUNCTIONS DESCRIBE THE VARIATION OF APPLIED LOADS
DURING THE SOLUTION, BY SCALING THE LOADVALUES GIVEN IN THE
DATA.  LOAD-VS-TIME FUNCTIONS MAY BE GENERATED AUTOMATICALLY
FROM THE OPTIONS BELOW ----

   (1) - STEP FUNCTION (CONSTANT)
   (2) - STEP FUNCTION OVER A SPECIFIED TIME INTERVAL
   (3) - RAMP FUNCTION
   (4) - TRIANGULAR PULSE
   (5) - USER DEFINED


LIST ELEMENT PRESSURE DATA ( Y , N ) ---Y

 E L E M E N T   P R E S S U R E   D A T A

   FIRST LAST INCR SURF CASE  PRESSURE
     21   30    1    6    1 -.10000E+01
```

```
                  ***** DATA CURVE OPTIONS *****
I                                   I
I     ( OPTION 1 )                  I       ( OPTION 2 )
I                                   I
XXXXXXXXXXXXXXXXXXXXXXXXXXXXX       XXXXXXXXXXXXXX
X                                   X            X
X                                   X            X
X                                   X            X
X----------------------------       X------------XXXXXXXXXXXXXXXXX

I                       X           I               X
I                   X               I            X     X
I               X       .
                                    
I               X                   I        X              X
I           X                       I      X                   X
I       X                           I    X                       X
I   X                               I  X                           X
I X       ( OPTION 3 )              I X      ( OPTION 4 )            X
X----------------------------       X--------------------------------X

( OPTION 5  =  USER - DEFINED DATA CURVE )

ENTER TIME FUNCTION TYPE FOR CURVE NO.   1 -3

PEAK LOADING VALUES ARE ASSUMED TO BE THOSE SPECIFIED IN THE
ORIGINAL DATA FILE.  DO YOU WISH TO SCALE THESE VALUES (Y/N) N

PEAK LOADING VALUES ARE ASSUMED TO BE THOSE SPECIFIED IN THE
ORIGINAL DATA FILE.  DO YOU WISH TO SCALE THESE VALUES (Y/N)N


L = (L)IVE ELEMENT SURFACE PRESSURES
D = (D)EAD LOADING

ENTER SURFACE PRESSURE TYPE ( L , D ) --L


****************************************
*                                      *
*        DATA GENERATION COMPLETE      *
*                                      *
****************************************


    STOP
    055500 MAXIMUM EXECUTION FL.
     1.338 CP SECONDS EXECUTION TIME.
COMMAND- REWIND,FDATA.
COMMAND- REQUEST,MYFILE,*PF.
COMMAND- COPYBF,DATA,MYFILE.
 ***EOI/EOF/EOR ENCOUNTERED-NO DATA COPIE
 D***
COMMAND- COPYBF,FDATA,MYFILE.
COMMAND- CATALOG,MYFILE,FOLDEDPLATEDATA,ID=BROCKMAN,RP=999
 INITIAL CATALOG
 CT ID= BROCKMAN PFN=FOLDEDPLATEDATA
 CT CY= 001 SN=AFFDL   0000003392 WORDS.
```

SECTION 8

REFERENCES

1. T. Harwiss, "Large Analyses in A. S. Computas," Finite Element News, pp. 6-9, September 1979.

2. PATRAN G User's Guide, PDA Engineering, Santa Ana, Ca., 1980.

3. IMPRESS Finite Element System User Manual, Third Edition, United Computing Systems, Inc., June, 1981.

4. R. A. Brockman, "MAGNA (Materially and Geometrically Nonlinear Analysis) Part I - Finite Element Analysis Manual," Air Force Wright Aeronautical Laboratories, Wright-Patterson Air Force Base, Ohio 45433, AFWAL-TR-82-3098, Part I, December 1982.

5. R. W. Stineman, "A Consistently Well-Behaved Method of Interpolation," Creative Computing, July, 1980.

6. R. E. McCarty, Flight Dynamics Laboratory, AFWAL/FIER, Wright-Patterson AFB, Ohio 45433 (private communication).

7. G. J. Peters, "Interactive Computer Graphics Application of the Parametric Bicubic Surface to Engineering Design Problems," in Computer-Aided Geometric Design, R. E. Barnhill and R. F. Riesenfeld, eds., Academic Press, 1974.

SECTION 9

APPENDICES

APPENDIX A

PREPROCESSOR FILE DESCRIPTIONS

The preprocessor described in this report deals with two internal forms and two external forms of finite element model data. These forms are discussed briefly in Section 6 - Model Data Files. Table A.1 describes the unformatted 2-D internal data file that is utilized for shell-like surface models defined via the programs CORGEN, AGRID and SPATCH described in Section 2. The EXPAND utility converts all shell surface model files to the standard unformatted 3-D internal data file illustrated in Table A.2. This file type is accessed by the preprocessor and the data translation programs. Once the user has defined the model, there exist two external file types for transferring data from the preprocessor to other programs or computer systems. Table A.3 describes the neutral data file format, which can be used to transfer data between the preprocessor and other programs, to transmit data between computer systems, or to archive modeling data in a form which can be listed on a terminal or line printer. The MAGNA input file, which is input to TRNSFR and output from REFMT, is described in detail in Reference [4].

TABLE A.1

SHELL SURFACE GEOMETRY FILE

File description: This is an unformatted (binary), sequential file of surface (2-D) model data output from the data entry programs AGRID, CORGEN and SPATCH. The numbering of node points and elements is sequential with numbering implicit.

Section	# Records	Variables	Description
I	1	NUMNP NUMEL	Number of node points Number of elements
II	NUMNP	COORD(4)	Cartesian coordinates and thickness for node point
III	NUMEL	NCON(9)	Nodal connectivity for element

TABLE A.2

THREE-DIMENSIONAL MODEL DATA FILE

File description: This is an unformatted (binary), sequential file of three-dimensional model data output from the data entry programs CREATE, EXPAND and IJKGEN and the interface routines (e.g. NEUTRAL, IMPRINT and TRNSFR .

Section	# Records	Variables	Description
I	1	NUMNP	Number of node points
		NUMEL	Number of elements
		NUMBC	Number of boundary conditions
II	NUMNP	NODNUM	Node point number (arbitrary)
		COORD(3)	Cartesian coordinates (X,Y,Z)
III	NUMEL	IELNUM	Element number (arbitrary)
		ITYPE	Element type code
		MATRL	Material type code
		IAXSET	Material axis code
		INTORD	Integration order
		IDUM(3)	Unused
		NCON(27)	Nodal connectivity for elements (variable 1-27 nodes solid)
IV	NUMBC	IBEG	First node
		IEND	Last node
		INCR	Node increment
		IDIR(3)	List of constrained directions
V	1	NUMNL	Number of nodal loads
VI	NUMNL	NBEG	First node
		NEND	Last node
		INCR	Node increment
		ICASE	Case number
		F(3)	Nodal force vector (X,Y,Z)
VII	1	NUMLE	Number of element loads
VIII	NUMLE	NBEG	First element
		NEND	Last element
		INCR	Element increment
		ISURF	Surface or edge number
		ICASE	Case number
		FORCE	Force per unit area/length
IX	1	NUMLC	Number of linear constraints
X	NUMLC	NODE	Node number
		XMULT	Multiplier for X-displacement
		YMULT	Multiplier for Y-displacement
		ZMULT	Multiplier for Z-displacement

TABLE A.3

NEUTRAL FILE DATA FORMAT

File description: This is a formatted (character), sequential file of three-dimensional model data output from the NEUTRAL program. Records are all 80 columns.

Section	# Records	Variable	Format	Description
I	1	NUMNP	I5	Number of node points
		NUMEL	I5	Number of elements
		NUMBC	I5	Number of boundary conditions
II	NUMNP	NODNUM	I5	Node point number (arbitrary)
		COORD(3)	E15.8	Cartesian coordinates (X,Y,Z)
III	NUMEL	IELNUM	I5	Element number (arbitrary)
		ITYPE	I5	Element type code
		MATRL	I5	Material type code
		IAXSET	I5	Material axis code
		INTORD	I5	Integration order
		IDUM(3)	I5	Unused
		NCON(27)	I5	Connectivity for elements (variable 1-27 nodes)
IV	NUMBC	IBEG	I5	First node
		IEND	I5	Last node
		INCR	I5	Node increment
		IDIR(3)	I5	List of constrained directions
V	1	RUMNL	I5	Number of nodal loads
VI	NUMNL	NBEG	I5	First node
		NEND	I5	Last Node
		INCR	I5	Node increment
		ICASE	I5	Case number
		F(3)	3E15.8	Nodal force vector (X,Y,Z)
VII	1	NUMLE	I5	Number of element loads
VIII	NUMLE	NBEG	I5	First element
		NEND	I5	Last element
		INCR	I5	Element increment
		ISURF	I5	Surface or edge number
		ICASE	I5	Case number
		FORCE	E15.8	Force per unit area/length
IX	1	NUMLC	I5	Number of linear constraints
X	NUMLC	NODE	I5	Node number
		XMULT	E15.8	Multiplier for X-displacement
		YMULT	E15.8	Multiplier for Y-displacement
		ZMULT	E15.8	Multiplier for Z-displacement

APPENDIX B

POSSIBLE DATA PATHS

Tables B.1 and B.2 contain a brief summary of the preprocessing programs and data file types referred to in this report. These, in conjunction with the flow chart in Figure 1.1, are intended to aid in determining the best path to follow through the system in preparing a finite element analysis model.

For a particular application, the type of data available for use should be compared with the data entry descriptions given in Table B.1. The type of data generated by any of the data entry or translation utilities can then be determined from Table B.2. The data type generated can next be compared with the input file types in Table B.2 to determine which utility (usually EXPAND or PREP) will be executed next. Once the model data is stored in the internal, three-dimensional file format used by PREP, the data path will generally take the form PREP --> REFMT --> MAGNA for analysis, or PREP --> NEUTRAL for the translation to the format required by other programs.

TABLE B.1

PREPROCESSOR MODULE FUNCTIONS

INITIAL DATA GENERATION		
AGRID	:	Definition of surface geometry via an arbitrarily arranged grid of points.
CORGEN	:	Lofting and/or direct data input of arbitrary surface geometry.
CREATE	:	Direct input and editing of coarse mesh geometry.
EXPAND	:	Expansion of surface geometry description into three-dimensional model form.
IJKGEN	:	Mesh generation on analytically-defined surfaces.
SPATCH	:	Model generation from surface 'patch' description.
INTERFACING MODULES		
IMPRINT	:	Conversion of IMPRESS preprocessor file to MAGNA preprocessor format.
NEUTRAL	:	Translation of model data between internal format and external (text) format.
TRNSFR	:	Conversion of MAGNA input to preprocessor format.
PREPROCESSING		
PREP	:	Model refinement, merging, plotting, properties definition, etc.
REFMT	:	Generate MAGNA input data deck.

TABLE B.2

PREPROCESSOR FILE TYPES

	FILE TYPE	TYPE FILE	OUTPUT FROM	INPUT TO
1	Internal 3-D Data File	Unformatted Sequential	CREATE TRNSFR IJKGEN PREP EXPAND NEUTRAL IMPRINT	PREP REFMT NEUTRAL
2	Internal 2-D Data File	Unformatted Sequential	CORGEN SPATCH AGRID	EXPAND
3	Material Properties Library File	Formatted Sequential	-	PREP REFMT
4	Neutral Data File	Formatted Sequential	NEUTRAL	NEUTRAL
5	MAGNA Input Deck	Formatted Sequential	REFMT	MAGNA TRNSFR

APPENDIX C

ACCESS PROCEDURES FOR PREPROCESSOR MODULES

Procedures for accessing the individual preprocessor modules are summarized in Tables C.1 and C.2 for quick reference. On CDC machines all of the programs may be accessed through a single CCL (CYBER Contro Language) command procedure, PREPROCESSORPROC; execution is initiated by entering a single BEGIN command as described in Table C.1. On the VAX 11/780, the DCL (Digital Command Language) procedure [MAGNA.RAB] CONTROL may be used to control execution, or a RUN command may be issued directly.

TABLE C.1

CDC ACCESS TO MAGNA PREPROCESSING SYSTEM

All preprocessor modules may be accessed through a single CCL procedure file, which is attached as follows:
ATTACH,P,PREPROCESSORPROC,ID=BROCKMAN,SN=AFFDL,MR=1.

Program Name	Operating Mode	Input File	Output File(s)	User Subroutines	Access
AGRID	Interactive	INGEOM	SURFAC	none	BEGIN,AGRID,P. CATALOG,SURFAC,filename.
CORGEN	Interactive	optional TAPE11 - TAPE20	TAPE10	none	[ATTACH,TAPExx,filename.] BEGIN,CORGEN,P. CATALOG,TAPE10,filename.
CREATE	Interactive	optional UNFMTO	UNFMT	none	[ATTACH,UNFMTO,filename.] BEGIN,CREATE,P. CATALOG,UNFMT,filename.
EXPAND	Interactive Batch	TAPE10	TAPE11	none	ATTACH,TAPE10,filename. BEGIN,EXPAND,P. CATALOG,TAPE11,filename.
IJKGEN	Interactive	none	UNFMT	USUB (SURFAC, CRDTRN)	[ATTACH,USUB,filename.] BEGIN,IJKGEN,P. CATALOG,UNFMT,filename.
IMPRINT	Interactive	TAPE10	TAPE9	none	ATTACH,TAPE10,filename. BEGIN,IMPRINT,P. CATALOG,TAPE9,filename.
NEUTRAL	Interactive	UNFMT FMTDAT	FMTDAT UNFMT	none	ATTACH,UNFMT,filename. BEGIN,NEUTRAL,P. CATALOG,FDATA,filename.
PREP	Interactive	TAPE10 - TAPE22	TAPE10 - TAPE22	none	ATTACH,TAPExx,filename(s). BEGIN,PREP,P. RETURN,TAPEyy,TAPEzz,... (unwanted files) REQUEST,A,*PF. REWIND,A,TAPEaa. COPYBF,TAPEaa,A. CATALOG,A,filename.
REFMT	Interactive	UNFMT	FDATA	none	ATTACH,UNFMT,filename. BEGIN,REFMT,P. CATALOG,FDATA,filename.
SPATCH	Interactive	TAPE50	SURFAC	USUB (UOPEN,UCLOSE UPATCH)	ATTACH,TAPE50,filename. BEGIN,SPATCH,P. CATALOG,SURFAC,filename.
TRNSFR	Interactive	FMTDAT	UNFMT	none	ATTACH,FMTDAT,filename. BEGIN,TRNSFR,P. CATALOG,UNFMT,filename.

TABLE C.2

VAX ACCESS TO MAGNA PREPROCESSING SYSTEM

All programs may be accessed through the CONTROL procedure by typing @[DIRECTORY]CONTROL, where the actual directory name ([DIRECTORY] is used in the Table) will be installation-dependent. Most programs allow the user to specify input and output files as they are required by the program. For these programs there is an indication of 'yes' for input and output files. All other programs will show the default file names the program will look for during execution.

Program Name	Operating Mode	Input File(s)	Output File(s)	User Subroutines	Access
CREATE	Interactive	optional	UNFMT.DAT	none	RUN [DIRECTORY]CREATE
EXPAND	Interactive	yes	UNFMT.DAT	none	RUN [DIRECTORY]EXPAND
IJKGEN	Interactive	none	UNFMT.DAT	USUB	RUN [DIRECTORY]IJKGEN
NEUTRAL	Interactive	yes	yes	none	RUN [DIRECTORY]NEUTRAL
PREP	Interactive	yes	yes	none	RUN [DIRECTORY]PREP
REFMT	Interactive	yes	yes	none	RUN [DIRECTORY]REFMT
SPATCH	Interactive	yes	yes	USUB	RUN [DIRECTORY]SPATCH
TRNSFR	Interactive	yes	yes	none	RUN [DIRECTORY]TRNSFR

APPENDIX D

USER SUBROUTINES FOR IJKGEN

The user subroutines for IJKGEN (UINPUT, SURFAC, and CRDTRN) have the following general forms:

```
      SUBROUTINE UINPUT
      COMMON <variable list>
          .
          .
      < user - written FORTRAN code to read or initialize
        variables in blank COMMON >
          .
          .
      RETURN
      END


      SUBROUTINE SURFAC ( I, J, K, ALPHA, BETA, ZETA )
      COMMON <variable list>
      COMMON / LIMITS /   IMAX ,    JMAX ,     KMAX ,
     +                  ALFALO ,  ALFAHI ,   BETALO ,
     +                  BETAHI ,  ZETALO ,   ZETAHI
          .
          .
      < user - written FORTRAN code to define the curvilinear
```

```
  coordinates ALPHA, BETA, and ZETA as functions of I,
  J, and K.  The generated coordinates should form a
  right-handed coordinate system but are otherwise
  arbitrary >
   .
   .
RETURN
END


SUBROUTINE CRDTRN ( ALPHA, BETA, ZETA, X, Y, Z )
COMMON <variable list>
COMMON / LIMITS /   IMAX ,    JMAX ,     KMAX ,
+                  ALFALO ,  ALFAHI ,   BETALO ,
+                  BETAHI ,  ZETALO ,   ZETAHI
   .
   .
< user - written FORTRAN code to define the transformation
  from curvilinear coordinates ALPHA, BETA, and ZETA to
  right-handed Cartesian coordinates X, Y, Z >
   .
   .
RETURN
END
```

Normally, UINPUT will not be used unless at least one of the other routines is also supplied to IJKGEN. When UINPUT is used to set the values of parameters in blank COMMON, these values will not be modified elsewhere within the program and may be used and/or modified as needed within SURFAC and CRDTRN.

The named COMMON block /LIMITS/ contains data concerning the limits of the mesh being generated, which is often useful within the user subroutines SURFAC and CRDTRN. The three integer values IMAX, JMAX, KMAX contain the maximum values of the generator indices (their lower limit is always one), and the remaining parameters describe the lower and upper limits of each of the curvilinear coordinates, which are requested as keyboard input during execution. While the data in COMMON /LIMITS/ is always accessible to the user routines, the data values should not be modified to ensure proper operation of the program.

APPENDIX E

USER SUBROUTINES FOR SPATCH

The three available user-written subroutines for SPATCH are:

- UPATCH - define surface patch coefficients and generation dat
- UOPEN - open and/or position input file
- UCLOSE - close input file

In most instances, only the first routine, UPATCH, will be needed to perform the data conversion.

The general form of UPATCH is as follows:

```
      SUBROUTINE UPATCH ( PATCH, THICK, NU, NV, ICOUNT, IEND )
      DIMENSION   PATCH ( 4, 4, 3 )
C
C    PATCH  =  Output array of patch coefficients, having the same form
C              as in equation (2.3).
C              PATCH (i,j,1) = coefficient array (4x4) for X-coordinate
C              PATCH (i,j,2) = coefficient array (4x4) for Y-coordinate
C              PATCH (i,j,3) = coefficient array (4x4) for Z-coordinate
C
C    THICK  =  Thickness value for the patch, for use in EXPAND
C
C    NU     =  Number of quadratic elements to be generated from this
```

```
C                surface patch, along the u-coordinate direction
C
C    NV     =  Number of quadratic elements to be generated from this
C                surface patch, along the v-coordinate direction
C
C    ICOUNT =  Sequence number for this patch.  ICOUNT is provided in
C                case the user routine must keep track of the number of
C                patches read.  When UPATCH is entered for the n-th time,
C                the value of ICOUNT = n, unless modified by the user.
C
C    IEND   =  End of data flag, to be set in UPATCH.  If no more data
C                exists, IEND should be set to 1 to inform SPATCH that
C                all of the data has been read.  If IEND=Ø on exit from
C                UPATCH, SPATCH will generate surface data from the
C                parameters PATCH, THICK, NU and NV.
C

        .
        .
   < user-written code >
        .
        .
      RETURN
      END
```

The UPATCH routine may read data from any type of file (random, sequential, formatted, or unformatted). For this reason, a second user-written routine, UOPEN, is provided to permit initialization of the file to be read. The form of UOPEN is simply

```
      SUBROUTINE UOPEN
          .
          .
      < code to initialize file >
          .
          .
      RETURN
      END
```

On CDC machines, the local file TAPE5Ø is reserved for use by UPATCH, and the UOPEN routine need only be provided if the surface patch data is stored on a random access file; in this instance, UOPEN may be used to initialize the random file by calling OPENMS, for example. Random file index keys may be declared in blank COMMON if desired, since SPATCH does not use unlabelled COMMON blocks. With the VAX version of SPATCH, UOPEN must be provided to open the file containing surface patch data.

UOPEN is called at the beginning of execution, before any data is requested from UPATCH. When the file to be read contains other data preceding the surface patch data, UOPEN can be used to position the file properly.

The third user routine, UCLOSE, is provided to permit closing of the user data file following execution. Normally, UCLOSE will not be required for either machine version (VAX, CDC), but is provided for use in exceptional circumstances. UCLOSE is called as the last step in execution; its form is

```
SUBROUTINE UCLOSE
    .
    .
< user code >
    .
    .
RETURN
END
```

APPENDIX F

MATERIAL PROPERTIES LIBRARY

The material properties library used by PREP (Section 4) and REFMT (Subsection 5.1) contains materials data for twelve classes of materia

ACRYL - Acrylics
ALUMI - Aluminum alloys
CASTI - Cast irons
COPPR - Copper-based alloys
GLASS - Glasses
MAGNS - Magnesium Alloys
NICKL - Nickel alloys
PLYMR - Polymeric and silicone materials
POLYC - Polycarbonates
STEEL - Carbon steels
STSTL - Stainless steels
TITNM - Titanium

Each material in the library is described by an accession number (five digits), a five-character class designation (e.g., ALUMI), a text description, and a list of numerical property values. Stored property information consists of:

- elastic modulus;
- Poisson's ratio;

- elastic shear modulus;
- mass density;
- coefficient of thermal expansion; and
- initial yield stress.

At present, the library does not contain material data for the plastic range of deformation (e.g., strain-hardening slope or full stress vs. strain curves), but allowance has been made for references to such tables, stored within the library. Two such table pointers are stored as part of the material description for each entry in the library.

*USGPO: 1983-659-062-902

END

FILMED